

N79-3162

NASA CR-159041

ATLAS – An Integrated Structural Analysis and Design System

Atlas User's Guide

Dr. Rodney L. Dreisbach

**Boeing Commercial Airplane Company
Seattle, Washington 98124**

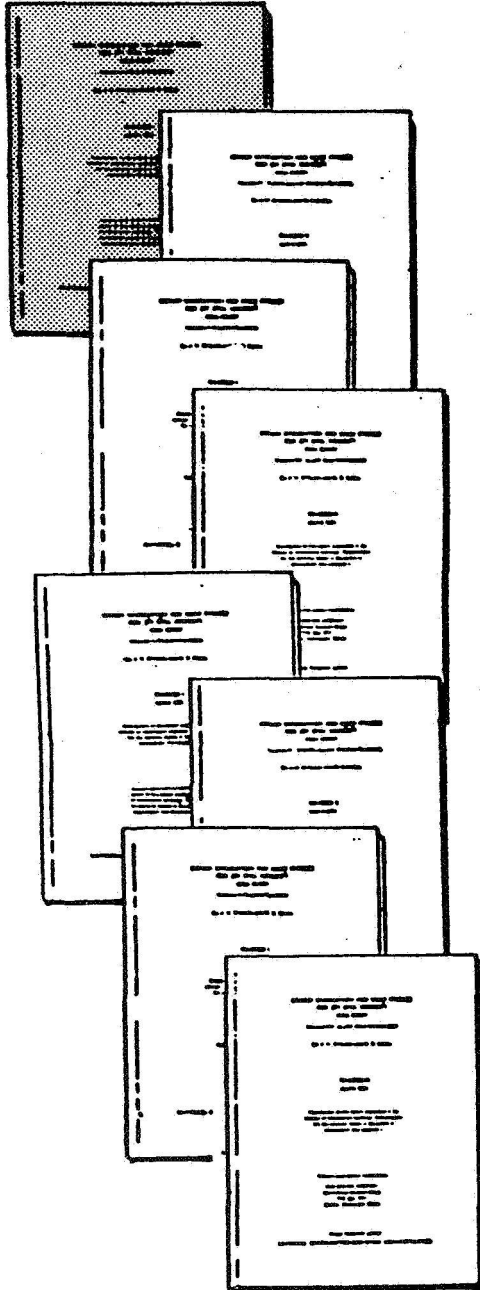
**Prepared for
Langley Research Center
under contract NAS1-12911**



National Aeronautics and
Space Administration

1979

ATLAS SYSTEM DOCUMENTATION



VOLUME I

ATLAS User's Guide

NASA CR-159041

VOLUME II

System Design Document

NASA CR-159042

VOLUME III

User's Manual-Input and Execution Data

NASA CR-159043

VOLUME IV

Random Access File Catalog

NASA CR-159044

VOLUME V

System Demonstration Problems

NASA CR-159045

VOLUME VI

DESIGN Module Theory

NASA CR-159046

VOLUME VII

LOADS Module Theory

Boeing Commercial Airplane Company

D6-25400-0101

VOLUME VIII

SNARK User's Manual

Boeing Computer Services

BCS-G0686

FOREWORD

Development of the ATLAS integrated structural analysis and design system was initiated by The Boeing Commercial Airplane Company in 1969. Continued development efforts have resulted in the release and application of several extended versions of the system to aerospace and civilian structures. Those capabilities of the ATLAS 4.0 version developed under the NASA Langley Contract No. NAS1-12911 include the following: geometry control, thermal stress, fuel generation/management, payload management, loadability curve generation, flutter solution, residual flexibility, strength design of composites, thermal fully stressed design, and interactive graphics. The monitor of this contract was G. L. Giles. The inertia loading capability was developed under the Army Contract No. DAAG46-75-C-0072.

This document is one volume of a series of documents describing the ATLAS System. The remaining documents present details regarding the input data and program execution, the program design and data management, the engineering method used by the computational modules, and system-demonstration problems.

The key responsibilities for development of ATLAS have been within the Integrated Analysis/Design Systems Group of the Structures Research Unit of BCAC and the ATLAS System Group of the Boeing Computer Services (BCS) Integrated Systems and Systems Technology Unit. R. E. Miller, Jr. was the Program Manager of ATLAS until 1976 after which K. H. Dickenson assumed this position. The current ATLAS System is the result of the combined efforts of many Boeing engineering and programming personnel. Those who contributed directly to the current version of ATLAS are as follows:

B. F. Backman	H. B. Hansteen	C. D. Mounier
G. N. Bates	B. A. Harrison	F. D. Nelson
L. C. Carpenter	J. M. Held	M. C. Redman
R. E. Clemmons	M. Y. Hirayama	R. A. Samuel
R. L. Dreisbach	J. R. Hogley	M. Tamekuni
W. J. Erickson	H. E. Huffman	G. von Limbach
S. H. Gadre	D. W. Johnson	S. O. Wahlstrom
F. P. Gray	A. S. Kawaguchi	R. A. Woodward
D. W. Halstead		K. K. Yagi

Principal contributors to the preparation of this document include: B. F. Backman, R. L. Dreisbach, S. H. Gadre, F. P. Gray, J. R. Hogley, and R. A. Samuel in the ATLAS Structures Research Group of the Boeing Commercial Airplane Company, Seattle, Washington.

ABSTRACT

This document describes some of the many analytical capabilities provided by the ATLAS Version 4.0 System in the logical sequence in which model-definition data are prepared and the subsequent computer job is executed. The example data presented in this document and the fundamental technical considerations that are highlighted can be used as guides during the problem solving process. This guide does not describe the details of the ATLAS capabilities, but provides an introduction to the new user of ATLAS to the level at which the complete array of capabilities described in the ATLAS User's Manual can be exploited fully.

The ATLAS System is operational on the Control Data Corporation (CDC) 6600/CYBER computers in the batch, on-line, interactive, and mixed computing environments. It is a modular system of computer codes with common executive and data-base management components.

An integrated software-system architecture is used to provide cost effective analyses of structures in a small or large problem environment. The system provides an extensive set of general-purpose technical programs with analytical capabilities including stiffness, stress, loads, mass, substructuring, strength design, unsteady aerodynamics, vibration, and flutter analyses. The stiffness finite-element structural-analysis approach is used wherein the physical properties of the structure are represented by a finite number of idealized elements.

The sequence and mode of execution of selected program modules are controlled by a common user-oriented language. Interactive user interfaces are provided for performing execution-control, data-file editing, and graphical-display of selected data. Communication of data between program modules is performed automatically by the ATLAS data-base manager, which can also be used to manipulate data for interfacing selected modules with external programs.

Model-definition input data are written in a problem-oriented language. Input-data generation options and input-data checks provided by the preprocessors minimize the amount of data and flowtime for problem definition/verification. Postprocessors allow selected data to be extracted, manipulated, and displayed via on-line or off-line prints and plots for monitoring and verifying problem solutions.

The design of the ATLAS System has been based primarily on the needs of the analyst in performing single or multidisciplined analysis/design tasks. From the technical and system reviews with the users, extended versions of the system have been released for general use. In the anticipation of extending the system capabilities, a high degree of flexibility and generality has been incorporated into many segments of the system.

CONTENTS

	Page
1.0 INTRODUCTION	1.1
2.0 ATLAS SYSTEM DOCUMENTATION	2.1
2.1 SYSTEM DESIGN DOCUMENT	2.1
2.2 USER'S MANUAL--INPUT AND EXECUTION DATA	2.1
2.3 RANDOM ACCESS FILE CATALOG	2.2
2.4 SYSTEM DEMONSTRATION PROBLEMS	2.2
2.5 LOADS MODULE THEORY	2.3
2.6 DESIGN MODULE THEORY	2.3
2.7 SNARK USER'S MANUAL	2.4
3.0 OVERVIEW OF ATLAS	3.1
3.1 ANALYTICAL CAPABILITIES	3.2
3.1.1 Analysis Control	3.2
3.1.2 Data Management	3.3
3.1.3 Technical Computations	3.3
3.2 FUNCTIONAL ORGANIZATION	3.8
3.2.1 Executive Modules	3.11
3.2.2 Data Base Management System	3.11
3.2.3 Computational Modules	3.12
4.0 PROBLEM-SOLVING USING ATLAS	4.1
4.1 DATA PREPARATION	4.1
4.2 JOB DECK PREPARATION	4.3
4.3 JOB PROCESSING	4.5
4.3.1 Data Preprocessing	4.6
4.3.2 Data Postprocessing	4.6
4.3.3 Solution Accuracy Checks	4.7
5.0 MODEL-DEFINITION DATA	5.1
5.1 DATA DECK STRUCTURE	5.3
5.1.1 Input Options and Format	5.7
5.1.2 Data Generation	5.9
5.1.3 Data Comments	5.12
5.2 STRUCTURAL MODEL	5.12
5.2.1 Reference Frames	5.13
5.2.2 Geometry and Nodes	5.16
5.2.3 Boundary Conditions	5.21
5.2.4 Stiffness Finite Elements	5.27
5.2.5 Fundamental Modeling Considerations	5.42
5.2.6 Solution Accuracy Checks	5.47
5.3 MASS DISTRIBUTION MODEL	5.49
5.3.1 Mass Finite Elements	5.50
5.3.2 Concentrated Masses	5.55

5.3.3	Fuel and Payload	5.56
5.3.4	Mass Modeling Considerations	5.59
5.3.5	Mass Options	5.60
5.4	STATIC LOADS	5.64
5.4.1	Support Displacements	5.65
5.4.2	Nodal and Element-Distributed Loads	5.65
5.4.3	Rotational Inertia Loads	5.65
5.4.4	Thermal Loads	5.66
5.4.5	ATLAS/FLEXSTAB Interfaced Loads	5.67
5.5	STRESS	5.68
5.5.1	Problem Formulation	5.68
5.5.2	Solution Accuracy Checks	5.69
5.5.3	Stress Postprocessing	5.70
5.6	BUCKLING	5.71
5.6.1	Problem Formulation	5.71
5.6.2	Solution Accuracy Checks	5.73
5.7	VIBRATION	5.74
5.7.1	Problem Formulation	5.74
5.7.2	Solution Accuracy Checks	5.76
5.8	FLUTTER	5.77
5.8.1	Problem Formulation	5.77
5.8.2	Modal Interpolation	5.80
5.8.3	Oscillatory Aerodynamic Forces	5.82
5.8.4	Residual Flexibility Effects	5.99
5.8.5	Combination and Interpolation of Airforce Matrices	5.99
5.8.6	Flutter Analysis	5.100
5.9	SUBSTRUCTURED MODELS	5.102
5.9.1	Substructured Stress Analysis	5.103
5.9.2	Substructured Vibration Analysis	5.105
5.9.3	Automated Interaction Criteria	5.106
5.9.4	Treatment of Loadcases	5.108
5.9.5	Treatment of Analysis Frames	5.108
5.9.6	Multi-Level Interaction	5.108
5.9.7	Recommended Practices	5.109
5.10	STRUCTURAL RESIZING	5.109
5.10.1	Fully Stressed Design (FSD)	5.112
5.10.2	Thermal Fully Stressed Design (TFSD)	5.114
5.10.3	Composite Optimization	5.116
5.10.4	Element Property Smoothing	5.118
6.0	EXECUTION-CONTROL DATA	6.1
6.1	CONTROL PROGRAM FUNCTIONS	6.2
6.2	PRINTED OUTPUT	6.7
6.2.1	Automatic Printout	6.7
6.2.2	Requested Printout	6.12
6.3	GRAPHICAL OUTPUT	6.12
6.3.1	Display Options	6.16
6.3.2	Interactive Graphics	6.27

6.4	INTERACTIVE PROCESSING	6.35
6.4.1	ATLAS Interactive Mode	6.37
6.4.2	EDIT Interactive Mode	6.37
6.4.3	EXECUTE Interactive Mode	6.39
6.4.4	Example Interactive Processing Session . .	6.39
7.0	JOB CONTROL CARDS	7.1
7.1	EXAMPLE JOB CONTROL DECKS	7.2
7.1.1	Batch Execution--Model and Execution Data Decks on Cards	7.2
7.1.2	Batch Execution--Model and Execution Data Decks in Permanent Files	7.2
7.1.3	On-Line Execution--Model and Execution Data Decks in Permanent Files	7.2
7.1.4	Using a Control Program in Absolute Form .	7.3
7.1.5	Job Execution Checkpoint	7.3
7.1.6	Job Execution Restart	7.4
7.1.7	Plot Data File Creation	7.5
8.0	EXAMPLE PROBLEMS	8.1
8.1	EXAMPLE 1: STRESS ANALYSIS	8.5
8.1.1	Description of Analysis	8.5
8.1.2	Execution-Control Data	8.5
8.1.3	Model Definition Data	8.6
8.1.4	Results	8.9
8.2	EXAMPLE 2: BIFURCATION BUCKLING	8.19
8.2.1	Description of Model	8.19
8.2.2	Description of Analysis	8.19
8.2.3	Execution-Control Data	8.20
8.2.4	Model-Definition Data	8.21
8.2.5	Results	8.23
8.3	EXAMPLE 3: WEIGHTS ANALYSIS	8.31
8.3.1	Description of Model	8.31
8.3.2	Execution-Control Data	8.32
8.3.3	Model-Definition Data	8.32
8.3.4	Results	8.35
8.4	EXAMPLE 4: VIBRATION ANALYSIS	8.39
8.4.1	Description of Model	8.39
8.4.2	Description of Analysis	8.39
8.4.3	Execution-Control Data	8.39
8.4.4	Model Definition Data	8.40
8.4.5	Results	8.42
8.5	EXAMPLE 5: FLUTTER ANALYSIS	8.49
8.5.1	Description of Model	8.49
8.5.2	Description of Analysis	8.49
8.5.3	Execution-Control Data	8.50
8.5.4	Model-Definition Data	8.51
8.5.5	Results	8.54
8.6	EXAMPLE 6: SUBSTRUCTURED STRESS ANALYSIS	8.64
8.6.1	Description of Model	8.64

8.6.2	Description of Analysis	8.64
8.6.3	Execution-Control Data	8.66
8.6.4	Model-Definition Data	8.68
8.6.5	Results	8.71
8.7	EXAMPLE 7: SUBSTRUCTURED VIBRATION ANALYSIS . .	8.86
8.7.1	Description of Model	8.86
8.7.2	Description of Analysis	8.86
8.7.3	Execution-Control Data	8.86
8.7.4	Model Defintion Data	8.87
8.7.5	Results	8.90
8.8	EXAMPLE 8: STRUCTURAL DESIGN	8.97
8.8.1	Description of Analysis	8.97
8.8.2	Execution-Control Data	8.98
8.8.3	Model-Definition Data	8.99
8.8.4	Results	8.102
9.0	SNARK LANGUAGE	9.1
9.1	CONTROL PROGRAM APPLICATIONS	9.1
9.1.1	Data Management	9.1
9.1.2	Matrix Arithmetic	9.2
9.2	EXTERNAL PROGRAM INTERFACES	9.5
9.2.1	Control Program Example	9.7
9.3	SPECIAL ANALYSIS CAPABILITY	9.11
9.3.1	Control Program Example	9.13
GLOSSARY		G.1
REFERENCES		R.1

FIGURES

No.	Page
3-1 ATLAS System Modular Design	3.10
4-1 ATLAS Job Deck Setup	4.4
5-1 Idealization of Real Structure	5.2
5-2 Model-Definition Data Deck Structure for an ATLAS Stress Analysis	5.6
5-3 Kinematic and [Force] Labels and Conventions	5.15
5-4 Example Use of Local Reference Frames	5.17
5-5 Nodal Reference Frame Coordinates	5.18
5-6 Example Nodal Data	5.20
5-7 Typical Geometry Components	5.22
5-8 Symmetric and Antisymmetric Displacements	5.25
5-9 Typical End-Force Releases for BEAM Elements	5.33
5-10 Example Data Records Used to Define BEAM Elements	5.34
5-11 A Warped (Distorted) Quadrilateral Plate	5.37
5-12 Element Stress Interpretation	5.45
5-13 Example Plate Element Grid	5.52
5-14 Fuel Tank and Cargo Hold Basic Shapes	5.57
5-15 Offset Beam Mass Lumping	5.63
5-16 The Normal to an Aerodynamic Surface	5.84
5-17 AF1 Aerodynamic Idealization	5.90
5-18 Placement of RH03 Downwash Points	5.98
5-19 A Substructured Model Interact Diagram	5.103
5-20 Example Substructured Model	5.104

5-21	Regional Composite Optimization	5.117
5-22	Strength Critical Laminate Screening	5.117
5-23	Composite Optimization Problem Solution Logic	5.119
6-1	Execution-Control Deck Echo and Compilation Printout	6.8
6-2	Example Printout of a Control Statement at Execution Time	6.10
6-3	Model-Definition Data Deck Echo	6.10
6-4	Partition 1,1 Row/Freedom Correspondence Table	6.13
6-5	Pivot-Decay Solution Accuracy Message	6.13
6-6	Point Plot of Nodes	6.19
6-7	Element Grid with Element Numbers	6.19
6-8	Exploded Geometry Plot	6.20
6-9	Deformed Geometry Plot Options	6.22
6-10	Example Scalar-Grid Plot	6.23
6-11	Contour Plot Options	6.25
6-12	Example Graph Plots	6.26
6-13	Example Matrix Data Plot	6.28
6-14	Graphics Function Menu Options	6.29
6-15	Typical GNAME Directory	6.29
6-16	Typical Plot ID Directory	6.29
6-17	Plot Transformation Menu	6.31
6-18	Example Use of the Plot Transformation Menu	6.31
6-19	Zoomed On-Line Display	6.34
6-20	Interactive Execution Control and Processing Functions	6.36
6-21	Data Deck for Example Interactive Session	6.40

6-22	On-Line Display of Initial Model Geometry	6.43
6-23	On-Line Display of Corrected Model Geometry	6.45
8-1	Geometry of the Wing Box Structure	8.2
8-2	Layout of the Structural Model	8.3
8-3	Wing Box Structural Model as Plotted Using the GRAPHICS Postprocessor	8.4
8-4	Plane Frame Model	8.19
8-5	Wing Panel Definitions	8.31
8-6	Doublet Lattice Box Grid	8.49
8-7	Substructured Wing Box Structural Model	8.65
8-8	Total Weight vs. Design Cycle Number	8.112
9-1	Schematic of ATLAS/FLEXSTAB System Interfaces	9.6
9-2	Procedural Outline for Control Program Coding	9.12

TABLES

No.	Page
3-1 ATLAS Preprocessors, Processors and Postprocessors . . .	3.14
5-1 Model-Definition Input Data for Standard Types of Analyses	5.4
5-2 The ATLAS Stiffness Elements	5.29
5-3 Characteristics of the ATLAS Stiffness Elements	5.30
5-4 The ATLAS Mass Elements	5.53
6-1 Summary of the ATLAS Execution Control Statements . . .	6.3
6-2 Processor/Postprocessor Executions for Standard Types of Analyses	6.6
6-3 Standard Label Subsets as Used to Create Data Plots . .	6.15
6-4 Technical-Data Plot Types	6.17
6-5 Summary of the ATLAS Text-Editor Requests	6.38
9-1 Summary of the SNARK Commands	9.3

1.0 INTRODUCTION

This User's Guide is designed to facilitate use of the ATLAS System for solving static and dynamic structural engineering problems. It presents some of the many options provided by ATLAS in the logical sequence in which model-definition data are prepared and the subsequent computer job is executed. The new user is encouraged not to be concerned about the information in later sections of this document prior to when it is required. The example input data and execution data presented in this document can be used as guides when using ATLAS during the problem solving process.

As the user becomes familiar with ATLAS, the more sophisticated features of the system will prove to be invaluable. The level of success in solving structural engineering problems is influenced not only by the analyst's experience, but also by his imagination and ingenuity as applied to new problems and solution methods. In dealing with the ever-increasingly complex structural configurations and environments, the user will find innumerable combinations of the ATLAS analytical options.

This guide provides an introduction to the new ATLAS user to the level at which the complete array of ATLAS capabilities, as described in the ATLAS User's Manual, can be exploited fully. Detailed descriptions of all model-definition input data and execution-control data for the ATLAS System are presented in the User's Manual which is designed as a reference for the familiar user. The user is encouraged to make reference to the other volumes of the ATLAS System documentation as progress is made through this guide.

The overall organization of the User's Guide begins with a summary of the contents of the current ATLAS documentation in section 2. Section 3 contains an overview of all the current analytical capabilities of ATLAS (Version 4.0) and a description of the functional organization of the system. Fundamental considerations that will aid the analyst in preparing model-definition data and an ATLAS job deck for execution are discussed in section 4, with a description of the automatic and user-selected solution accuracy checks performed during job processing. Descriptions of the model-definition data and data deck organization for structural and mass models, static load definition, stress, buckling, vibration and flutter analyses, substructured models, and automated structural resizing are presented in section 5.

Descriptions of the user-defined execution-control data for batch, and on-line interactive execution of ATLAS, including the user-control of printed and plotted data and the interactive graphics capabilities, are presented in section 6. The computer operating-system job control cards used for processing an ATLAS job are described in section 7.

A number of different types of ATLAS analyses, using a common structural model, are presented in section 8, along with example execution-control decks, data decks, and selected output.

The SNARK matrix-interpretive language, as used by ATLAS and which is available for general use in writing special execution-control decks, is described in section 9, along with usage guidelines and example applications.

A glossary of common terms closely associated with the use of ATLAS is provided at the end of this document.

The User's Guide does not present details of all the ATLAS System capabilities, but provides an introduction to the engineer planning to use ATLAS, and directs him to the other ATLAS documentation. As the engineer becomes familiar with using ATLAS, more extensive reference will be made to the ATLAS User's Manual in applying the many analytical options and alternatives provided by ATLAS, as well as its more sophisticated features.

2.0 ATLAS SYSTEM DOCUMENTATION

The User's Guide is one volume of a series of documents that describe the ATLAS Version 4.0 System. Descriptions of the input data and program execution, the system-demonstration problems, the program design and data management, and the engineering method used by the computational modules are presented in the other documents. When the engineering theory used by a computational module is well documented by other publications, they are referenced in the ATLAS User's Manual. Otherwise, the engineering method used by a computational module is presented in a separate ATLAS document. The contents of each ATLAS document, with the exception of the User's Guide, are summarized below.

2.1 SYSTEM DESIGN DOCUMENT (Volume II)

This document describes the overall system design, the design of the individual program modules, and the ATLAS System library routines. The system design is discussed in terms of its architecture, executive functions, data base structure, and operational procedures for program maintenance and modification. Code descriptions, use of common blocks, and file usage by each program module are presented. The ATLAS system library routines are described to the level that these general purpose algorithms can be used in the creation of special-purpose execution-control decks. Extensive use of comment cards included in the program codes fully complement the contents of this document, which is designed primarily for use by an ATLAS System programmer.

2.2 USER'S MANUAL -- INPUT AND EXECUTION DATA (Volume III)

This document presents detailed descriptions of all model-definition input data and execution-control statements provided by the ATLAS System. This manual is designed as a reference book rather than a text that is to be used during problem solving. The format of this document is as follows:

- a) Section 10 presents an overview of the ATLAS System functions, its architecture and its computational capabilities.
- b) Section 11 describes the general deck-setup requirements, the CDC-System job-control cards, and the resource requirements for job processing.
- c) Section 100 presents general rules for preparing model-definition input data. This section describes the structure of an input data deck, the input data formats, the general capabilities provided for data generation,

and definitions of the terminology used for mathematical models.

- d) The remaining 100-series sections present detailed descriptions of the input data corresponding to each of the technical-data preprocessors. These sections are ordered alphabetically according to the preprocessor names for convenient referencing.
- e) Section 200 presents the method and general rules for executing the ATLAS System computational modules. This section describes the structure of an execution-control deck, the executive statement formats, cataloged control procedures, and the execution checkpoint/restart facilities.
- f) The remaining 200-series sections present detailed descriptions of the executive-control statements and run-time parameters associated with each of the technical processor and postprocessor modules. These sections are ordered alphabetically according to the module names for convenient referencing.
- g) The appendices include libraries of the structural and mass finite elements, descriptions of the executive-control procedures provided for standard types of analyses, sample ATLAS job-deck setups, descriptions of the standard interfaces of ATLAS with external computer programs, and descriptions of the ATLAS interactive EDIT module commands for data-file definition and modification.

The model-definition input data, the execution control statements, and the SNARK language commands for the ATLAS System are summarized in a User's Pocket Manual.

2.3 RANDOM ACCESS FILE CATALOG (Volume IV)

This document presents a complete catalog of the random access data files used by the ATLAS System. Each of the data matrices output to these files by the program modules is described in detail. This document is used primarily by the ATLAS user or by an ATLAS System programmer to manipulate data directly within the ATLAS data base.

2.4 SYSTEM DEMONSTRATION PROBLEMS (Volume V)

This document describes a set of problems that demonstrate the analysis and design capabilities provided directly by the ATLAS System as well as the analytical capabilities provided by

the interfaces of ATLAS with external computer programs. The discussion of each problem includes the following:

- a) Description of the problem to be solved, the analysis to be performed, the mathematical model, and the load environment.
- b) Discussion of the ATLAS results. Comparisons are made to theoretical solutions, where possible.
- c) Listings of the ATLAS execution-control decks and model-definition data.

All typical analyses performed by ATLAS and practically all of the ATLAS input/execution options are exercised by these problems.

2.5 LOADS MODULE THEORY (Volume VI)

The engineering theory underlying the operation of the ATLAS LOADS module is presented in this document. The equations used to calculate statically-equivalent or consistent nodal forces from input data such as pressure distributions and rotational inertia are presented for the different types of structural finite elements.

2.6 DESIGN MODULE THEORY (Volume VII)

The engineering design theory underlying the automated structural-resize methods provided by the ATLAS DESIGN module is presented in this document. The methods, typical applications, and limitations associated with the fully stressed design, the thermal fully stressed design, and the regional composite-structure optimization algorithms are described. A discussion of the convergence characteristics of the fully stressed design theory is also included in this document.

2.7 SNARK USER'S MANUAL (Volume VIII)

This document describes the syntax of the SNARK user-oriented language that provides standard matrix and scalar mathematical capabilities, as well as matrix-data and blank-common computer-core management functions.

The different types of SNARK commands can be grouped according to their function.

- a) Data Manipulation
- b) Random and Sequential File Manipulation
- c) Input/Output
- d) Matrix Arithmetic
- e) Programming Aids

Use of SNARK is required only by the sophisticated ATLAS user or by an ATLAS System programmer to perform the following:

- a) Manipulate data directly within the ATLAS data base to perform special analytical computations that are not provided directly by the ATLAS system, or to interface data with external computer programs via the ATLAS Control Program.
- b) Design, develop and integrate new computational capabilities within the ATLAS System.

3.0 OVERVIEW OF ATLAS

The problem-solving analytical capabilities of ATLAS and the functional organization of the ATLAS System are summarized in this section.

The ATLAS System is operational on the Control Data Corporation (CDC) 6600/CYBER computers in the batch, on-line, interactive, and mixed computing environments. It is a modular system of computer codes with common executive and data-base management components.

An integrated software-system architecture is used to provide cost effective analysis of structures in a small or large problem environment. The system provides an extensive set of general-purpose technical programs with analytical capabilities including stiffness, stress, loads, mass, substructuring, strength design, unsteady aerodynamics, vibration, and flutter analyses. The stiffness finite-element structural-analysis approach is used wherein the physical properties of the structure are represented by a finite number of idealized elements.

The sequence and mode of execution of selected program modules are controlled by a common user-oriented language. Interactive user interfaces are provided for performing execution-control, data-file editing, and graphical-display of selected data. Communication of data between program modules is performed automatically by the ATLAS data-base manager, which can also be used to manipulate data for interfacing selected modules with external programs.

Model-definition input data are written in a problem-oriented language. Input-data generation options and input-data checks provided by the preprocessors minimize the amount of data and flowtime for problem definition/verification. Postprocessors allow selected input and calculated data to be extracted, manipulated, and displayed via on-line or off-line prints and plots for monitoring and verifying problem solutions.

The design of the ATLAS System has been based primarily on the needs of the analyst in performing single or multidisciplined analysis/design tasks. Based on periodic reviews of the technical requirements, extended versions of the system have been released for general use. In the anticipation of extending the system capabilities, a high degree of flexibility and generality has been incorporated into many segments of the system.

3.1 ANALYTICAL CAPABILITIES

ATLAS provides the user with three major types of analytical capabilities.

- a) Analysis Control
- b) Data Management
- c) Technical Computations

Each of these capabilities is described further in sections 3.1.1 through 3.1.3, respectively.

3.1.1 Analysis Control

The user has complete control of the analysis functions and problem-solution steps to be performed by ATLAS. These functions are defined by the executive control data which are written in the ATLAS user-oriented control language. Directives supplied by the execution-control data are used to define the following:

- a) Sequence of Computations
- b) Execution Parameters
- c) Management of Analysis Results
- d) Scheduled Restart of Problem Execution
- e) Contingencies when Data Errors are Encountered

The execution-control directives can also be used to perform special analytical computations that are not provided directly by ATLAS, manipulate ATLAS data, and manage data for interfacing the ATLAS System modules (sub-programs) with other external computer programs (see secs. 9.2 and 9.3).

At execution time, a Control Program is created automatically from the user-supplied execution-control data. Execution of the Control Program and the ATLAS System can be performed in an interactive, on-line, batch, or mixed computing mode. Any ATLAS System module can be executed, as needed, while solving a problem interactively, by defining the necessary execution-control statements using a computer terminal during job processing. Other interactive control features include the on-line definition of selected data plots and the display of selected plots on the screen of a graphics terminal.

3.1.2 Data Management

Management of all data within the system is performed automatically. It is, however, the user's responsibility to manage the following types of data:

- a) Model-Definition Input Data
- b) Execution-Control Input Data
- c) Problem Execution Checkpoint-Restart Data

Input-data files and control files can be defined, modified and saved by using the ATLAS interactive text editor. Furthermore, checkpoint-restart procedures are provided for performing stepwise problem solutions. By using these data management facilities, the engineering user is not burdened by data communication problems when solving single or multidisciplined problems. More of the analyst's time can thereby be spent on the engineering aspects of the problem solving process.

3.1.3 Technical Computations

The many analytical capabilities of ATLAS can be grouped as follows:

- a) Linear Stress Analysis
- b) Bifurcation Buckling Analysis
- c) Weights Analysis
- d) Vibration Analysis
- e) Flutter Analysis
- f) Substructured Analyses
- g) Structural Resizing

The technical computations are based on the following matrix equation which defines the equilibrium condition for a linear structure.

$$M*A + K*U = P$$

where M is the mass matrix, K is the stiffness matrix, and P is the loads matrix for the overall (gross) structure. Matrices "A" and "U" are the acceleration, and displacement matrices, respectively, for the nodal points used to describe the kinematics of the structural model. Creation of the gross stiffness matrix, for example, is performed by assembling (merging) the stiffness matrices calculated for the individual finite elements.

Variations of the foregoing equation that are used to solve different technical problems are

$$\begin{aligned} K*U &= P \text{ for static stress analysis,} \\ (K + E*G)*Q &= 0 \text{ for buckling analysis, and} \\ (K - E*M)*Q &= 0 \text{ for vibration analysis.} \end{aligned}$$

Matrices E, Q, and G denote eigenvalues, eigenvectors, and geometric stiffnesses, respectively. The underlying method of finite element analysis, in all cases, is known by the following different names:

- a) Displacement Method--U
- b) Stiffness Method--K
- c) Equilibrium Method--P

The foregoing groups of ATLAS analytical capabilities can either be combined or further subdivided, depending on the type of problem being solved, the selected method of analysis, and the desired end results. Therefore, the following outline is presented to summarize the various ATLAS technical-analysis capabilities.

Data Management and Execution Control

- A data-base manager
- A user-oriented execution-control language
- Catalogs of execution-control procedures
- Interactive, on-line and batch-mode processing
- Interactive data-file and control-file editor
- Job execution checkpoint-restart procedures
- Numerous problem-solution accuracy checks

Utility Matrix Algebra

- A user-oriented language for performing matrix and scalar mathematics
- A module for assembling elemental stiffness, mass, loads, and displacement matrices
- A module for solving out-of-core systems of linear symmetric equations
- A module for adding and multiplying out-of-core matrices

Data Preprocessing

- Free-field, input-data format
- Model-definition data written in a problem-oriented language; many data default values are provided

- Common, automatic data-generation options for all data types
- Automatic generation of nodes, element grids, loads, etc.
- Extensive number of warning and error diagnostics

Data Postprocessing

- Extract selected subsets of data for print/plot displays
- User-selected data printouts
- User-selected on-line and off-line plots
 - Orthographic, pictorial geometry plots
 - Data displays superimposed on element grids
 - Isocurve contour plots
 - X-Y graphs
- Interactive graphics

Geometry Modeling

- Lofting of three-dimensional structural components
- Maximum of 4095 nodes per data set or substructure
- Local rectangular, cylindrical, and spherical reference frames for node definition and for structural response

Boundary Conditions (BC)

- Constraints on selected nodal freedoms
- Symmetric and antisymmetric options
- Elastic supports
- Maximum of 10 different BC stages

Structural Modeling

- Maximum of 32 767 elements
- Nodal freedoms with no stiffness are automatically ignored
- Library of elements
 - Axial ROD and general BEAM
 - Membrane PLATE and bending GPLATE elements with orthotropic capability
 - Built-up SPAR and COVER elements
 - Family of 3-D isoparametric BRICK elements
 - Grounded SCALAR spring (elastic support)
 - Shear rod (SROD) and shear panel (SPLATE)
 - Composite-material plate (CPLATE) and built-up composite CCOVER elements

Multilevel Substructuring

- No limit on number of interact levels
- Automatic management of substructure-interact data
- Capability for stress, mass, and vibration analyses

Applied Static Loads

- Nodal loads
- Element distributed loads
- Thermal loading
- Rotational inertia loads
- Specified displacements
- Loadcase superposition
- No limit on number of loadcases

Linear Stress Analysis

- Displacement formulation
- Superposition of displacements and stresses
- Freebody--internal nodal forces on elements
- Equilibrium checks
- Stress contour plots

Bifurcation Buckling Analysis

- Maximum of 400 degrees of freedom
- Geometric stiffnesses for ROD, BEAM, PLATE, GPLATE, and BRICK elements
- Mode-shape plots

Strength Resizing

- Fully-stressed design
 - Panel buckling interaction
 - Geometric and margin-of-safety constraints
 - Thermal effects
- Smoothing of resized element properties
- Margin-of-safety plots

Structural Optimization

- Regional optimization of composite structures

Mass Analysis

- Maximum of 100 mass/weight distribution conditions
- Maximum of 32 767 nonstructural mass elements
- Library of elements
 - ROD and general BEAM
 - PLATE elements
 - Built-up SPAR and COVER elements
 - Concentrated SCALAR masses
- Detailed weight statements
- Diagonal, nondiagonal, and Guyan-reduced mass matrices
- Panel-weight matrices
- Fuel and payload management
- Fuel and payload loadability diagrams

Vibration Analysis

- Normal modes and frequencies
- Generalized mass and stiffness matrices
- Maximum of 400 degrees of freedom
- Mode-shape plots

Flutter Analysis

- Assumed-pressure function and doublet-lattice methods for subsonic compressible flow
- Strip-theory method for subsonic incompressible flow
- Mach-Box method for supersonic flow
- Residual structural-flexibility effects
- Mode interpolation functions
 - Surface spline
 - Motion axis and motion point
 - Polynomial
 - Beam spline
- Automated V-g and "matched point" solution options
- V-g and V-f flutter-solution plots

ATLAS Interfaces with External Programs

- ATLAS/FLEXSTAB--Interface ATLAS structural and mass data with FLEXSTAB for performing aeroelastic and elastic stability analyses; Interface FLEXSTAB steady-state loads with ATLAS for performing stress analysis and structural-design functions.
- ATLAS/NASTRAN--Interface ATLAS input data to NASTRAN and interface NASTRAN input data to ATLAS
- Interface geometry input data for the NASA-LRC aerodynamic configuration programs with ATLAS

3.2 FUNCTIONAL ORGANIZATION

The overall objective during the design and development of the ATLAS computer program has been to provide an integrated structural analysis and design system for the synthesis of aerospace vehicles in a timely, thorough, and cost-effective manner. The system design has been based primarily on the convenient, user-selection of system functions as required for performing either single or multidisciplinary analysis/design tasks. Continued emphasis has been directed toward the development of a system that:

- a) Provides a common executive module for convenient and versatile user-controlled technical-analysis flow and control of design cycles
- b) Provides a common data base that eliminates duplicate input data preparation, and centralizes the control of design data
- c) Provides data-management algorithms for convenient interfacing of the computational modules with external computer programs
- d) Uses data preprocessing and data generation codes to minimize the amount of input data, and the time required for data preparation and debugging
- e) Provides advanced engineering methodologies equally useful for performing design tasks in a small or large problem environment
- f) Uses automated structural-sizing algorithms to minimize the amount of hand-sizing of structural members, particularly during preliminary design studies
- g) Provides data postprocessing codes to extract, manipulate, and display (print/plot) selected data for monitoring analysis/design activities
- h) Provides interactive capabilities for defining and editing data files, for executing selected system modules, and for generating on-line and off-line graphical displays
- i) Provides an open-ended, low-cost admission of new program codes

The architecture developed for ATLAS is a modular system of overlaid program codes with common executive and data-base management components as illustrated in figure 3-1. Each module performs a well-defined engineering, mathematical, or clerical task. This modular design supports the foregoing system attributes and allows for effective development and maintenance of the system. Furthermore, through centralized management of the program modules, the reliability of the aggregate code is increased.

User interfaces with ATLAS are through the model-definition data and the ATLAS execution-control data (see fig. 3-1). The model data define the problem to be solved, whereas the control data specify which analysis functions are to be performed.

ATLAS has been designed to handle effectively those problems that require a small number or a very large number of degrees of freedom. Generally, the only limitations on problem size are those imposed by practical considerations of job-execution time and by the ultimate capacity of the auxiliary storage devices of the computer installation. ATLAS automatically performs many of the mathematical calculations by out-of-core processing as required for large problems. A detailed description of the system design is presented in the ATLAS "System Design Document."

The primary functional components of ATLAS, which provide the analysis control, data management, and technical computational capabilities, are as follows:

- a) Executive Modules
- b) Data Base Management System
- c) Computational Modules

Each of these components is described further in sections 3.2.1 through 3.2.3, respectively.

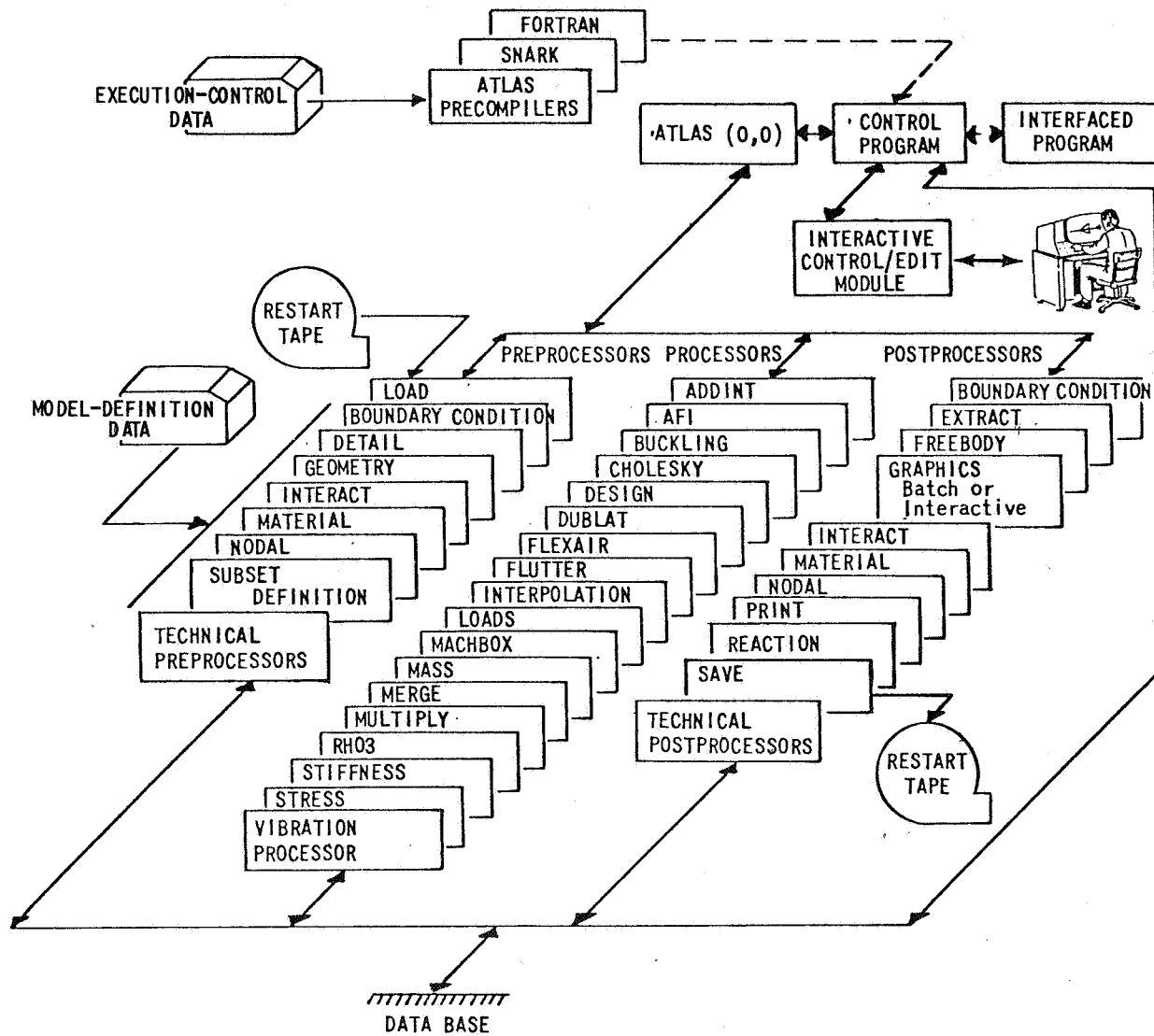


Figure 3-1. ATLAS System Modular Design

3.2.1 Executive Modules

The executive modules support the analysis control and data communication functions of the system. The executive modules and their basic functions are as follows:

- a) Precompilers -- Translate the user-oriented ATLAS and SNARK executive-language directives defined by the execution control data into equivalent FORTRAN statements. The resulting FORTRAN code is compiled at execution time to create an ATLAS Control Program module.
- b) Control Program -- Control the sequence and mode of execution of selected computational modules as specified by the user-provided execution-control data. Each execution-control statement initiates one or more steps in solving the problem defined by the model-definition data.
- c) Interactive Executive -- Interpret module execution-control directives that are input using the keyboard of a terminal during interactive processing, and perform interactive text editing of data files.
- d) ATLAS (0,0) Overlay -- Monitor the execution of computational modules per instructions from the Control Program module.

The executive modules, as illustrated in figure 3-1, allow ATLAS to be executed in a batch, on-line, interactive, or mixed computing environment.

Further descriptions of the ATLAS execution control data (execution directives) are presented in section 6.

3.2.2 Data Base Management System

Automatic transmission of data from one module to another is accomplished primarily by the use of named, random-access disk files. All input data interrogated by the preprocessors are stored in one file, whereas the data generated by a processor are stored in a separate file that is reserved for that module. Any of the data stored in the ATLAS data base can be accessed by any one of the computational modules.

The names of all the data files and data matrices associated with the ATLAS System are predefined in the codes. Options are provided, however, for user-specification of certain data matrix names (User Matrices) for greater versatility of data management during problem solving. Detailed descriptions of the ATLAS data files and data matrices are presented in the ATLAS "Random Access File Catalog."

The SNARK software package, an integral component of the ATLAS System, is used by the ATLAS modules to transfer data matrices directly from/to the data files to/from data arrays in core. SNARK is also used by the ATLAS modules to manage the blank-common computer core (the primary work area) during the computational processes. The SNARK library routines for management of data matrices and blank common, and the user-oriented SNARK language for performing matrix and scalar mathematics, can be used in defining ATLAS execution-control directives. These SNARK functions allow convenient access and manipulation of ATLAS data via a Control Program to perform special, user-defined computations, or to interface data with computer programs that are external to ATLAS. Further descriptions of SNARK usage are presented in section 9.

3.2.3 Computational Modules

The three types of ATLAS computational modules and their basic functions are as follows:

- a) Preprocessors -- Read, decode and interrogate the model-definition data; generate data based on a minimal number of input parameters; load the problem-execution restart data to resume processing.
- b) Processors -- Perform technical numerical computations
- c) Postprocessors -- Extract, format and display (print/plot) input data and analysis results; save the problem data for restart of problem solution by a subsequent job.

A technical processor performs a task that is related to a particular engineering theory or discipline. The STIFFNESS processor, for example, contains the code that represents the finite-element structural theory used in ATLAS. This module computes the stiffness and stress matrices for the finite elements used to define the structural model.

Certain processors are utility in nature in that they are used to perform general-purpose, normally out-of-core, mathematical operations. Examples of such operations are the solution of sets of linear equations performed by the CHOLESKY module, and the matrix addition and multiplications performed by the MULTIPLY module.

Table 3-1 contains a summary of all the ATLAS computational modules and their technical functions. Each module is identified by a different name. Generally, there is a preprocessor and a print postprocessor associated with each technical processor. That is, a preprocessor reads and interprets the set of data corresponding to a particular technology, whereas a postprocessor generates formatted printout of user-selected data corresponding to a certain technology.

Certain preprocessors and postprocessors are utility in nature in that they are not associated with a particular processor. These modules, shown separately in figure 3-1, are used to perform the specialty functions noted in table 3-1, as needed. Examples of such operations are the identification of subsets of nodes and elements in structural and mass models performed by the SUBSET DEFINITION preprocessor, and the on-line and off-line plotting of selected input and calculated data using the GRAPHICS postprocessor.

Table 3-1. ATLAS Preprocessors, Processors and Postprocessors

MODULE NAME				TECHNICAL FUNCTION
	PREPROCESSOR	PROCESSOR	POSTPROCESSOR	
ADDINT			● ●	Add and/or interpolate with respect to reduced frequencies those generalized airforce matrices generated by AF1, DUBLAT, FLEXAIR, MACHBOX, RH03, or by previous execution of ADDINT
AF1	●	●	●	Define aerodynamic model; Calculate subsonic, incompressible-flow aerodynamic loads for FLUTTER; Strip theory method
BC (BOUNDARY CONDITION)	●		●	Define boundary conditions for structural model
BUCKLING		●	●	Calculate bifurcation buckling loads and mode shapes
CHOLESKY		●		Solve systems of linear symmetric equations
DESIGN	●	●	●	Define structural resize data; Perform regional optimization of composite structures; Resize structural models based on a fully-stressed design, thermal and local-buckling effects, and geometric and margin-of-safety constraints
DETAIL	●			Define finite-element cross-section shapes for thermal gradients and stiffener locations for plate-element local buckling
DUBLAT	●	●	●	Define aerodynamic model; Calculate subsonic compressible-flow aerodynamic loads for FLUTTER; Doublet-lattice method
EXTRACT			●	Extract selected problem-definition and analysis data from the primary ATLAS data-base for GRAPHICS
FLEXAIR		●	●	Calculate generalized airforce matrices that include flexibility effects of truncated structural modes
FLUTTER	●	●	●	Define structural damping; Modify and solve the flutter equations
FREEBODY			●	Print internal nodal forces acting on selected finite elements
GEOMETRY	●			Define, generate, and interrogate three-dimensional geometry components for structural and mass models
GRAPHICS			●	Generate online and offline plots of data selected via EXTRACT; Batch and interactive-graphics modes of execution
INTERACT	●		●	Define substructure interaction problem
INTERPOLATION		●		Define mode-shape interpolation functions for AF1, DUBLAT, FLEXAIR, MACHBOX and RH03
LOAD	●			Load previously-generated problem data from SAVE to restart execution
LOADS	●	●	●	Define static loads; Calculate nodal loads due to inertial forces, pressure gradients, and thermal gradients; Cumulate static loads and specified displacements
MACHBOX	●	●	●	Define aerodynamic model; Calculate supersonic-flow aerodynamic loads for FLUTTER; Mach Box method
MASS	●	●	●	Define mass model that may complement STIFFNESS; Define fuel and payload management data; Generate mass matrices and detailed weight statements for primary and secondary structure, fuel, and payload
MATERIAL	●		●	Define finite-element material property data and design-allowable stresses for STIFFNESS, STRESS, MASS, and DESIGN
MERGE		●		Assemble stiffness, mass, loads, and displacement matrices
MULTIPLY		●		Add and multiply matrices
NODAL	●		●	Define local coordinate systems and nodal data for STIFFNESS and MASS models
PRINT			●	Print system-generated matrices
REACTION			●	Print reaction forces and load-reaction equilibrium checks for structural model
RH03	●	●	●	Define aerodynamic model; Calculate subsonic compressible-flow aerodynamic loads for FLUTTER; Assumed pressure modes method
SAVE			●	Save problem data for execution checkpoint; Use LOAD to restart execution
STIFFNESS	●	●	●	Define structural model; Generate elastic-stiffness, geometric-stiffness, and stress matrices for elements
STRESS	●	●	●	Assemble nodal displacements; Calculate element stresses and internal nodal forces; Superimpose displacements and stresses
SUBSET DEFINITION	●			Define subsets of nodes and elements in STIFFNESS and MASS models for subsequent processing; Define subsets of data-component labels for EXTRACT
VIBRATION		●	●	Calculate natural vibration frequencies and mode shapes in addition to generalized mass and stiffness matrices

4.0 PROBLEM-SOLVING USING ATLAS

The following information is presented in this section.

- a) Fundamental considerations in preparing model-definition data for an adequate mathematical model.
- b) The general deck setup requirements for an ATLAS job.
- c) Discussion of the automatic and user-selected data quality checks performed during job processing.

4.1 DATA PREPARATION

The following data preparation steps are suggested as a guide to be followed during the creation of a mathematical model of the physical problem to be solved.

- a) Develop a thorough understanding of the physical problem.
- b) Evaluate the available information concerning the physical problem.
- c) Determine what results are required.
- d) Establish a plan for the data preparation tasks.
 - Coordinate the plan with the individuals or groups of people that will contribute to the problem solving process. A multidisciplinary analysis of a complex structure, for example, may require the coordination of many people so that a properly phased and cost-effective analysis/design can be performed.
 - Identify any special modeling problems associated with the structure, special materials, mass, loads, and boundary conditions. Are there advantages of using a substructured model?
 - Select an appropriate method of analysis and establish a work breakdown of analytical tasks to be performed.
 - Develop "road maps" for performing the analysis and for verifying the idealized structural model as it relates to the real structure.

- Identify which units of measurement are to be used.
- e) Define the structural and/or mass finite element model. This fundamental task is one of the most important tasks that must be performed during the problem definition phase of work.
- Establish the geometry of the physical problem. Can any of the facilities provided for definition of three-dimensional geometry components be used effectively? Identify any local coordinate systems to be used in defining the geometry.
 - Establish the type of finite elements and finite element grids to be used in modeling distributed physical properties of the structure. What are the primary load paths? Can the symmetry properties of the structure and load environment be exploited? Adequate modeling requires the engineering user to have an understanding of the mechanics of the selected structural finite elements, the requirements for adequate modeling of the load environment, and the anticipated structural response.
 - Define the networks of nodes and elements for the model by transforming the actual hardware into a mathematical model. Are there advantages of using the data generation facilities? Node and element numbering, in addition to the identification of material and geometric element properties, are established during this step. Orderly arrangements of node and element numbers allow for more convenient interpretation of the calculated data.
- f) Define any local analysis frames to be used, and the kinematic boundary conditions for the nodes.
- Identify the coordinate systems to be used in describing the nodal kinematics.
 - Identify the kinematic conditions. What nodal degrees of freedom should be supported (displacements specified), or retained for generation of reduced matrices?
 - If a substructured model is to be used, establish the relationship between the substructure components. How shall the substructures interact and how many levels of interaction should be used

for effective problem solving? What types of parametric studies are anticipated?

- g) Define the loads acting on the structural model. What type of load environment must be accommodated? Are the loads distributed over the surface of the model or are they conveniently lumped at selected nodes?

Answers to some of the foregoing questions are dependent strictly on the physical problem to be solved. Some of the answers are learned primarily from experience gained in using finite element methods to solve a variety of practical structural problems. Fundamental considerations that will aid the analyst in preparing an adequate mathematical model, as well as how the fundamental capabilities of ATLAS can be used to solve typical problems, are presented in the remaining sections of this document. In the end, the engineering user is responsible for the development of a properly-discretized mathematical model, and is responsible for the method of analysis performed by using computer programs.

4.2 JOB DECK PREPARATION

The primary components of an ATLAS job deck in the order in which they are generally prepared for job processing are as follows:

- a) ATLAS Model-Definition Data Deck
- b) ATLAS Execution-Control Data Deck
- c) Control-Card Deck for the CDC 6600/CYBER

The different types of input data can be defined either by using cards or by using a computer terminal. Some or all of the data decks, for example, can be prepared on-line and stored in disk files or magnetic tape files and then accessed, as required, in lieu of using data cards. In describing the structure of the various data blocks and the job-execution control statements, reference is made primarily to the card input mode for clarity purposes only. Therefore, the term "deck" refers to a certain block of data, and not necessarily a card deck. Furthermore, the term "card" refers to a line of data, and not necessarily a physical card.

The first major task during the problem solving process is to define the mathematical model of the physical problem by creating a model-definition data deck. The format of model-definition data and an overview of the ATLAS modeling capabilities are described in section 5. Detailed descriptions of all input data are presented in the User's Manual.

The second major task is to define the execution-control data to specify the method of analysis, the sequence of computations, and the printed/plotted displays of selected data to be generated. These options are summarized in section 6. Detailed descriptions of all execution-control statements are presented in the User's Manual.

The third task is to define the job control cards for the CDC 6600/CYBER computer system on which the job deck will be executed. These cards are required to access the ATLAS program, assign disk and/or tape files as required for special data management (e.g., load data files for job-execution restart or manage off-line plot files), and provide accounting information for the computer operating system. The control cards used for typical ATLAS jobs are described in section 7 and are illustrated by the example problems in section 8.

The three decks are stacked in the sequence shown in figure 4-1 for job processing. The end-of-record cards are used to separate the decks, whereas the end-of-file is used to terminate the job deck.

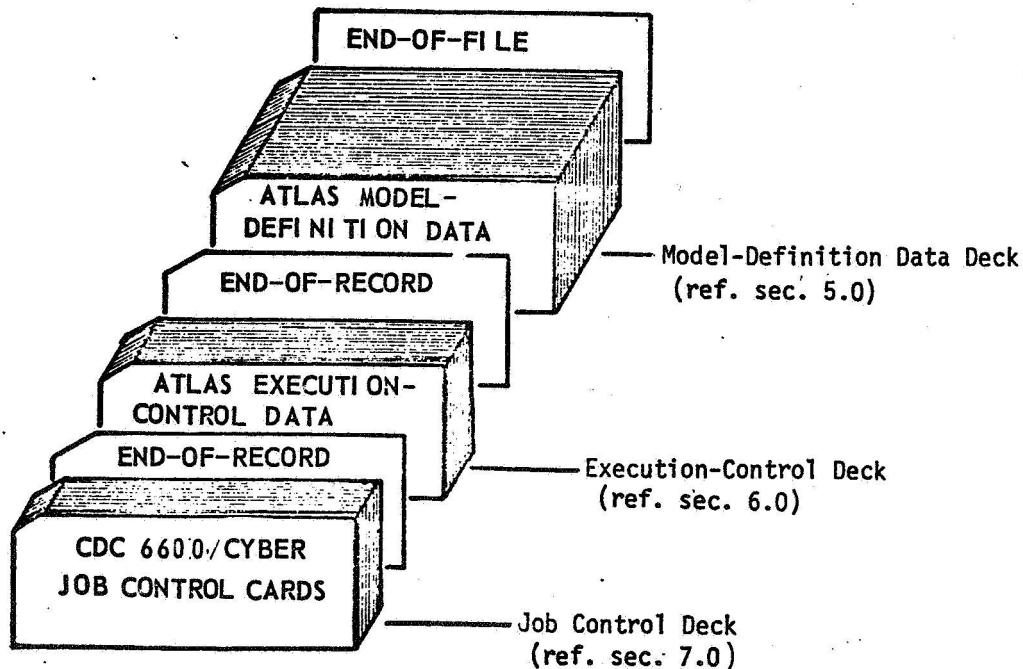


Figure 4-1. ATLAS Job Deck Setup

4.3 JOB PROCESSING

An ATLAS job can be executed in an interactive, on-line, batch, or mixed computing mode. Generally, small problems are processed most conveniently by executing the entire job in the interactive or on-line mode. Solutions to large problems are handled typically by performing selected preprocessing and postprocessing activities in the interactive or on-line mode, and the remaining computational tasks in the batch mode. The criteria used to select job processing modes are schedules, budget availability, and the maximum allowable computer-core allocation that can be requested for on-line job execution.

The first phase of job processing should always include checking the validity of the mathematical model. Although an extensive number of data checks are performed automatically by the preprocessors when the data deck is read, use of the convenience features provided by the PRINT and GRAPHICS postprocessors should be exploited to help check the data integrity. These facilities are particularly valuable during the data preparation and verification steps in the solution of large problems. Printout and plots of selected regions of the finite-element model and the deformed structure, for example, are much more convenient to use than many pages of printed data for the entire model.

During execution, ATLAS attempts to trap all anomalies and provides a self-explanatory message to the user when one occurs. When the code detects an ambiguity in the data which can be resolved without user interaction, a warning message is issued and the job execution proceeds uninterrupted. However, when a system error occurs, or when a fatal inconsistency is detected in the data or in the execution logic, an error message is issued. In this case, only the execution directives included in a user-prepared "error procedure" within the ATLAS control deck are processed prior to terminating the job (see sec. 6). Typically, an "error procedure" is used to save the problem data for subsequent restart of job processing at an intermediate solution step. In all cases, the word "WARNING" or the word "ERROR" is included in the diagnostic message issued to the output file.

For the more complex and costly analyses, checkpoints should be identified at intermediate steps in the solution sequence. Again, the features provided by the PRINT and GRAPHICS postprocessors should be used in conjunction with the solution-accuracy checks to help qualify the results. Use of the execution restart facilities (see sec. 6) allows job processing to begin at a selected checkpoint, subsequent to examination of the intermediate results.

4.3.1 Data Preprocessing

All of the model-definition data are read and interpreted by a single READ INPUT statement in the ATLAS execution-control deck (see sec. 6 for further details). During the data preprocessing, an extensive number of checks on the quality of the model input data are performed automatically. All of the data are preprocessed, even if errors are encountered. In this manner, multiple data errors can be detected and corrected prior to re-execution of the job.

4.3.2 Data Postprocessing

Formatted printout and graphical displays of selected input data and analysis results are generated only as requested by the user via appropriate ATLAS execution-control statements (see sec. 6 for further details). No fatal errors are issued during any of the postprocessing activities if data anomalies are detected; only warning conditions are identified. This convention is based on the premise that errors encountered during postprocessing do not necessarily mean that the job is fruitless and as such, job processing is continued.

Many of the input/output quantities associated with an ATLAS job are referenced to the nodes and elements used to define the mathematical model. For example, geometry, node loads, displacements and lumped masses are associated with nodes; stresses, distributed loads, etc., are associated with elements. The SUBSET DEFINITION preprocessor allows the user to identify groups of the nodes, stiffness elements and mass elements used to define a model. Particular data-component labels identifying which data quantities are to be extracted for a node/element subset can also be defined as label subsets.

Data associated with the subsets can then be extracted and manipulated by other ATLAS modules. This capability allows, for example, the printing and plotting of the node geometry and element-definition data associated with user-selected regions. Visual examination of selected parts of the model can thus be effected. Furthermore, interpretation and evaluation of the output data associated with selected subsets can be performed. For example, printed or plotted subsets of displacements, stresses and vibration mode shapes can be examined by use of this capability. All ATLAS graphical displays are created by using subsets defined by SUBSET DEFINITION input data.

Subset-definition data records allow the following types of manipulations to be performed:

- a) Node Subset Definitions
- b) Ordered Node Subset Definitions
- c) Element Subset Definitions
- d) Label Subset Definitions
- e) Extraction of Subsets from Other Subsets
- f) Definition of Subsets via Subset Combinations
- g) Subset Modifications (Exclusions)
- h) Node and Element Subsets Defined via Isolation of Geometric Regions

Further descriptions of the subset-definition capabilities and the format of the SUBSET DEFINITION input data are presented in section 156 of the User's Manual.

4.3.3 Solution Accuracy Checks

The model-definition input data used to idealize the physical problem are transformed automatically into systems of simultaneous equations. Solutions to these equations should be checked for correctness, primarily to see how well the mathematical model performed during the solution steps. If a solution has been generated, are the results reasonable? If the results are not reasonable or if the equations can not be solved uniquely, what modifications must be made to the mathematical model to overcome these dilemmas?

In addition to the extensive number of input data checks performed by the various ATLAS modules, many solution-accuracy checks are either performed automatically or are provided as execution options. Structural-model quality checks performed during the problem solution process are described in section 5.2.6, whereas those solution quality checks pertaining to specific types of analysis are described elsewhere in section 5.

5.0 MODEL-DEFINITION DATA

The first phase of the problem-solving process requires the distributed physical properties of the structure and its load environment to be approximated by discrete quantities, thereby making the physical problem amenable to mathematical analysis. During data preparation, therefore, the distributed characteristics of geometry, structural materials, and environment are transformed by the analyst into a set of ideal characteristics that are used to establish the model-definition data. These data, as input to the computer program, are converted by the program into a mathematical model characterized by a system of equations that is also solved by the program.

The creation of an accurate mathematical model of the physical problem, often called the process of idealization, requires the engineer to have an understanding of the mechanics of the selected structural materials and finite elements, the requirements for adequate modeling of the load environment, and the anticipated structural response. These criteria are essential for a successful analysis/design, particularly for complex structures and environments. In the end, it is the engineer's responsibility to develop a properly discretized mathematical model, and to interpret properly those results calculated by the computer program. Figure 5-1 illustrates the essence of transforming the real structure into an idealized structural model for automated analysis/design.

The components of an ATLAS mathematical model are defined by the model-definition data deck. Each component, referred to as an input data set, describes a particular characteristic of the model. Example data sets are those used to describe the geometry, boundary conditions, material properties, stiffness distribution, mass distribution, and the load and temperature distributions. The overall structure of an ATLAS model-definition deck is described in section 5.1.

Principal factors that should be considered during various types of modeling and problem solving are presented in the following sections. The discussions refer primarily to the model-definition data, however, those data that are input by the ATLAS execution-control statements, as related to the problem formulation process, are also described when necessary.

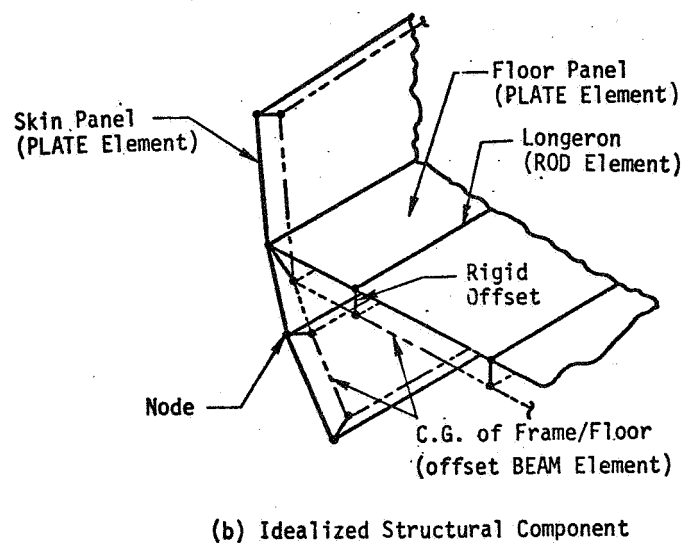
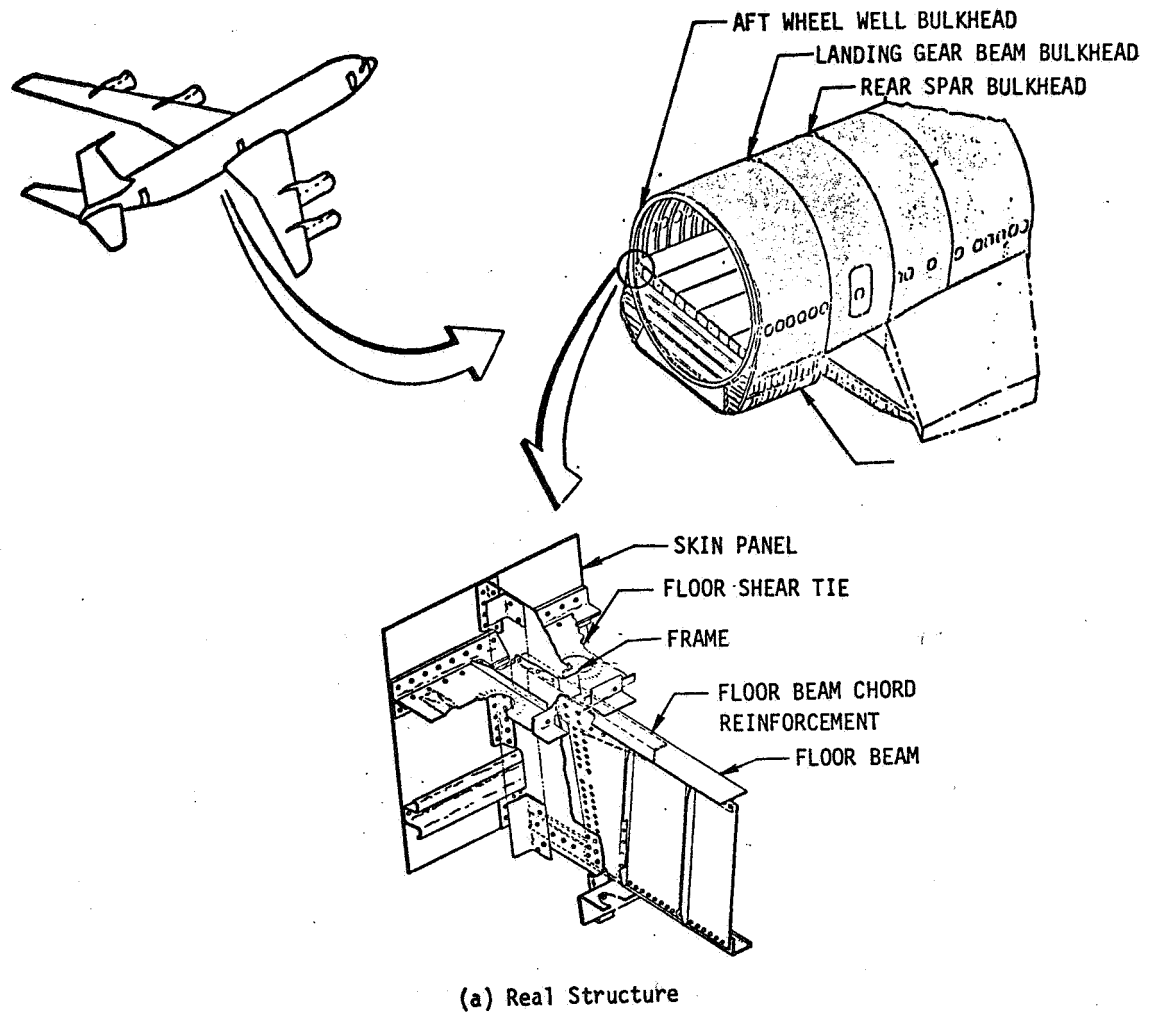


Figure 5-1. Idealization of Real Structure

<u>Section</u>	<u>Description</u>
5.1	Data Deck Structure
5.2	Structural Model
5.3	Mass Distribution Model
5.4	Static Loads
5.5	Stress
5.6	Buckling
5.7	Vibration
5.8	Flutter
5.9	Substructured Models
5.10	Structural Resizing

The reader who is only interested in selected types of modeling need only read the corresponding sections, in addition to section 5.1, and skip to section 6.

5.1 DATA DECK STRUCTURE (User's Manual; sec. 100)

An ATLAS model-definition data deck (see fig. 4-1), is composed of one or more input data sets. Each data set (a block of information) corresponds to a certain technology, and is read by one of the ATLAS data preprocessors. An input data set and its corresponding preprocessor are identified by the same name. Thus, the LOADS data set is read by the LOADS preprocessor.

The order in which the input data sets are stacked within a data deck for standard types of analyses, as shown in table 5-1, corresponds to the natural sequence followed during preparation of model-definition data. Thus, for example, when a structural model is being created, the NODAL (geometry) data are generally prepared first, followed by the BC (boundary condition) data and the STIFFNESS (finite-element) data. The BC data must follow the NODAL data for this example, but the STIFFNESS data set could be input first. Optional data-set input orders are described in the User's Manual.

Only those data sets that are required to define the problem to be solved need be included in the data deck. Those data sets which are required, and those data sets which are optional for standard types of analyses are identified in table 5-1. If the current computer job is an execution restart from a checkpoint established by a previous job, the checkpoint data (problem data saved from the previous job) must be provided in addition to any new data sets to be used in conjunction with the previous mathematical model. Execution checkpoint/restart facilities are described in section 6.

Table 5-1. Model -Definition Input Data for Standard Types of Analyses

Typical Stacking Sequence of Input Data Sets	Required and Optional Input Data Sets ①							
	Stress Analysis		Weights Analysis		Vibration Anal.		Flutter Analysis	
Name	Required	Optional	Required	Optional	Required	Optional	Required	Optional
GEOMETRY		*		*		*		*
NODAL	1	✓	1	✓	1	✓	1	✓
BC (BOUNDARY CONDITION)	2	✓		*	2	✓	2	✓
MATERIAL		*		*		*		*
STIFFNESS	3	✓		*	3	✓	3	✓
MASS			2	✓	4	✓	4	✓
SUBSET DEFINITION		*		*		*	5	✓
DETAIL		*						
INTERACT (SUBSTRUCTURE)		*		*		*		*
LOADS	4	✓						
STRESS		*						
DESIGN		*						
AF1							6	✓
DUBLAT							6	✓
MACHBOX							6	✓
RH03							6	✓
FLUTTER								*

①

- A check (✓) identifies a data set that is required for the corresponding type of analysis.
- An asterisk (*) identifies a data set that may be used to define the problem for the corresponding analysis type. Any one or combination of these data sets may be input.
- Any one or combination of AF1, DUBLAT, MACHBOX, and RH03 data sets may be input in any order.
- All of the data sets shown in the table or any combination thereof may be included in a data deck. Only those data needed by the execution-control statements specified in the Control Deck, however, are used during job processing.

The basic component of an input data set is a data record and its data items. Their relationship in a data deck is defined as follows:

- a) Data Deck -- One or more data sets
- b) Data Set -- A block of input data that is associated with a particular technology. A data set may include one or more data subsets.
- c) Data Subset -- A logical group of data within a data set. A data subset (and a data set) is composed of many data records.
- d) Data Record -- A record commonly implies one data card (one line of data), although it may require two or more cards. Each data record is composed of one or more data items.
- e) Data Item -- Either a number or an alphanumeric word (character string)

Figure 5-2 illustrates the relationship of these components within a data deck for an ATLAS stress analysis. For this example, the following data sets are illustrated.

- a) NODAL
- b) BC (Boundary Condition)
- c) STIFFNESS
- d) LOADS

The first record of a data deck can be either BEGIN PROBLEM DATA or the first "BEGIN Set-Name DATA" record of a data set (BEGIN NODAL DATA for this example). The last record of a data deck, however, must be an END PROBLEM DATA record. During job processing, the entire data deck is interrogated by a single READ INPUT statement in the execution-control deck (see sec. 6).

Each data set begins with a record of the form "BEGIN Set-Name DATA" and ends with an "END Set-Name DATA" record. Furthermore, an input data subset is identified by a record of the form "BEGIN Subset-Name DATA" and is terminated by its "END Subset-Name DATA" record. For the example shown in figure 5-2, a STIFFNESS data subset is identified by the record BEGIN ELEMENT DATA, and a LOADS data subset is identified by the BEGIN NODAL LOAD DATA record. Each word and number specified by the data records is referenced as a data item.

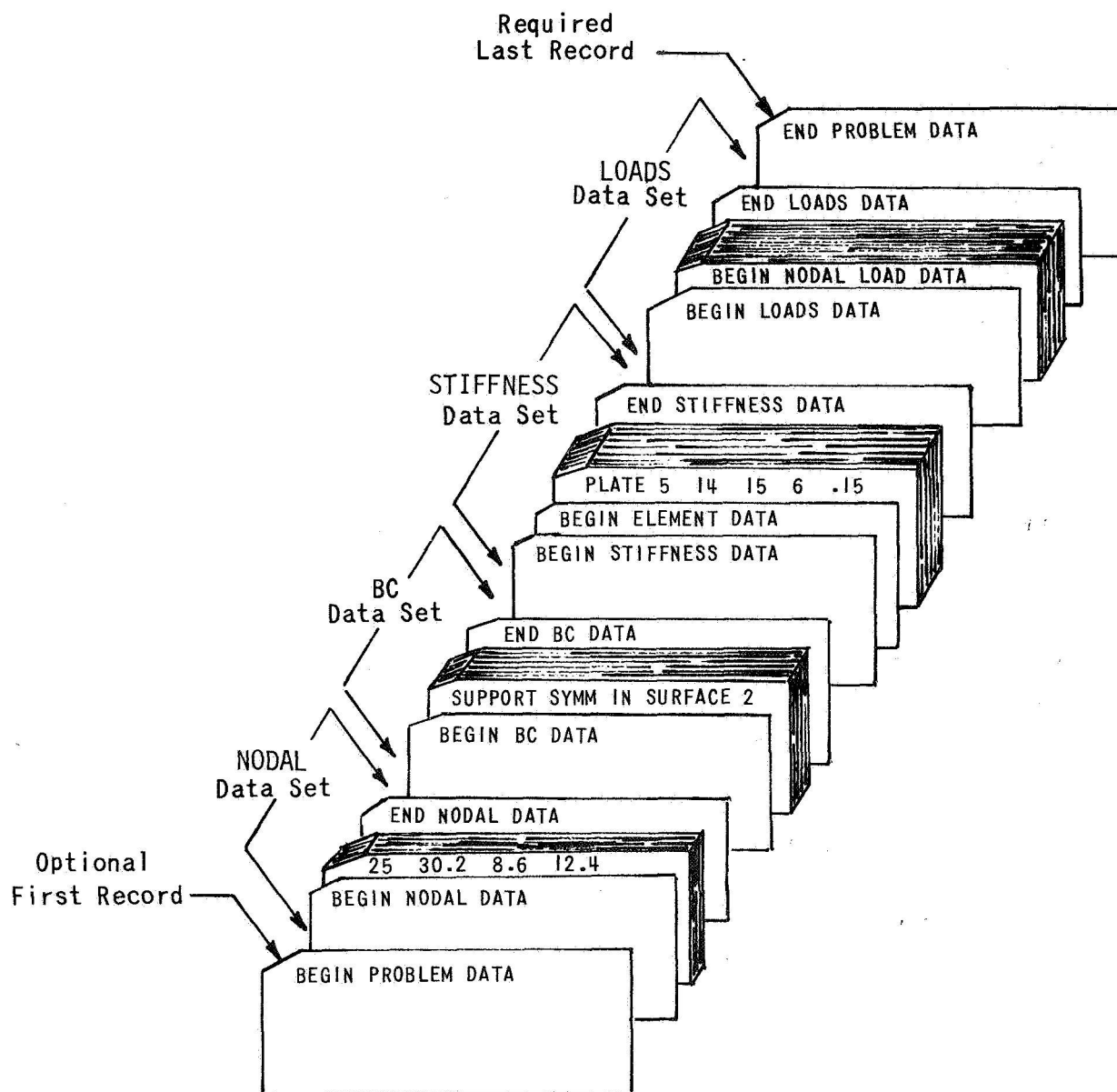


Figure 5-2. Model-Definition Data Deck Structure for an ATLAS Stress Analysis

General rules concerning input data formats for all data records, data items, and data comments are described in the following sections.

5.1.1 Input Options and Format

In describing the structure of the input data, reference is made primarily to the card input mode for clarity purposes only. Instead of using data cards, for example, other storage media such as disk files created by an interactive terminal can be used. Thus, the term "card" refers to a line of data, and not necessarily a physical card. Furthermore, the term "deck" refers to a certain block of data, and not necessarily a physical card deck.

All data items are input in a problem-oriented language using a free-field format with blanks, commas, colons, semicolons, or card boundaries as item separators (delimiters). All 80 columns of a card can be used to input a sequence of data items. Two forms of record input, referenced as MODE1 and MODE2, can be used.

- a) MODE1 -- The end of a data record is identified either by the / symbol or the \$ symbol. Information following the / symbol are treated as data deck comments (see sec. 5.1.3), whereas information following the \$ symbol are assumed to be data items for the next data record. A sequence of data items may continue onto as many cards as required to complete a data record.
- b) MODE2 -- Functions the same as MODE1 except the right-hand boundary of a card also acts as a record terminator, and a plus sign (+) must be used as the last character on a card when the record continues onto the next card.

The input of four data records is shown below to illustrate the two input formats. Each line denotes a card image.

Example MODE1 Input:

```
*/ MODE1 /  
BEGIN STIFFNESS DATA $ BEGIN  
ELEMENT DATA / DATA COMMENTS CAN BE INPUT AFTER THE SLASH SYMBOL.  
PLATE 5 14 15 5 .15 /  
PLATE 8, 16, 20,22, .20 /
```

Example MODE2 Input:

```
*/ MODE2 /  
BEGIN STIFFNESS DATA $ BEGIN +  
ELEMENT DATA / DATA COMMENTS CAN BE INPUT AFTER THE SLASH SYMBOL.  
PLATE 5 14 15 6 .15 /  
PLATE 8, 16, 20,22 .20 /
```

Both input-format modes can be used to define data records in a data deck. The initial input mode is set to MODE1 or MODE2 by the READ INPUT statement in the ATLAS execution-control deck (see sec. 6). The input format can then be changed anywhere within the data deck by use of one of the following input records:

```
*/ MODE1 /  
*/ MODE2 /
```

The input mode specified by the READ INPUT statement remains effective until it is changed by one of these records. Data records following one of these types or records are interpreted in the corresponding mode. The last input format specified by a record of this type remains effective until another record of this type is input.

The format of each data item is identified as one of the following:

- a) Decimal Number -- A positive or negative number input either with a decimal point or by use of the FORTRAN E-Format
- b) Integer Number -- A positive or negative number without a decimal point
- c) Alphanumeric Word -- An item with at least one nonnumeric character that can not be interpreted as a "decimal" or an "integer" number
- d) Alphanumeric Text -- Text strings may include any character except # (a quotation mark when using a terminal). A text string is initiated by the two characters *# and is terminated by the # character.

User-defined alphanumeric words are used, for example, as loadcase labels, material reference codes, and element-property reference codes. Other alphanumeric words, referenced as key-words, must be input exactly as shown in the detailed record descriptions presented in the User's Manual. Example key-words are illustrated in figure 5-2, which are primarily the words used to define the data-set and data-subset separator records.

Alphanumeric text, which is a sequence of user-defined characters, is used primarily for data-set related titles that are displayed in printout of the corresponding input and calculated data.

As noted in table 5-1, the use of many data sets is optional in defining a mathematical model for a certain type of analysis. Many data subsets are also identified as being optional. Input of these data is not required if the corresponding default data values are acceptable for definition of the problem. Additional default values and optional formats are provided for many of the data items. If the default value of a data item is acceptable for definition of the problem data, the corresponding item need not be input. Selection of the default values for the many required quantities characteristic of finite-element analysis has been based primarily on the frequency of user requirements during model-definition. This approach helps to minimize the amount of input data that the user must prepare, and yet provides the options and flexibility that are required for modeling a wide variety of problems.

The standard units of measurement used for ATLAS model-definition data are length in inches, force and weight in pounds, and temperature in degrees Fahrenheit. Compatible units of other input items are noted in the detailed descriptions of the data records in the User's Manual. The user may, however, choose to perform an analysis using any system of units that is desired, provided that consistency of units is ensured throughout the analyses performed by the selected computational modules.

5.1.2 Data Generation

One-dimensional and two-dimensional data generation capabilities are provided for convenient definition of strings and grids of nodes, elements, boundary conditions, loads, mass panels, etc. Use of these options reduces the amount of required user-supplied model-definition data.

The two types of data generation capabilities inherent to ATLAS are:

- a) Specialized Options -- Automatic generation of input data associated with a particular data preprocessor. Typical examples are the generation of nodes and coordinates, elements, and the overall geometry data required for finite-element modeling, as well as the carefully-selected default values for many of the required quantities. Another example of specialized data generation is the ATlist "a TO b BY c" string-generation option provided by many of the preprocessors.

The generated sequence begins with "a" and ends with "b" where the "n" intermediate values are generated by adding n*c to "a". Thus, the sequence "2,4,6,8,10" can be input as "2 TO 10 BY 2" and the sequence "T1A02,T1A05,T1A08,T1A11" could be input by the "T1A02 TO T1A11 BY 3" character string.

- b) Standard Options -- Automatic generation of items within data records, or generation of complete data records. Each of these options is controlled by an asterisk-type data item which can be used in any input data record. All types of data items except alphanumeric text can be generated by use of these features, as described below, where N is an unsigned integer.

- 1) *N -- A data item of the form *N denotes that the next N data items in a record are identical to the corresponding N items in the preceding record.
- 2) ** -- Double asterisks denote that all of the remaining items in a record are identical to the corresponding items in the preceding record.
- 3) *=N -- A data item of the form *=N denotes that the next N items in a record are identical to the immediately preceding item in that record.
- 4) *=N±STEP -- A data item of the form *=N±STEP, where STEP is an unsigned numeric item (integer or decimal), denotes that the next N items in the record are to be generated by consecutively incrementing the immediately preceding item in that record by ±STEP.
- 5) *+N -- A data record beginning with these characters denotes that the next N records are to be generated from the preceding record by adding an item of the *+N record to the corresponding item of the immediately preceding record. A zero (integer) item is used in an *+N record to indicate that an alphanumeric word in this record is identical to the corresponding alphanumeric word in the preceding record. Any combination of the preceding asterisk items, 1 through 4 above, may be used in an *+N record.

NOTE: An item in an *+N record that is to be added to a corresponding item in the preceding record (see 5 above) and the STEP item (see 4 above), must meet certain requirements. The item must be a decimal number if the

corresponding item is a decimal number, whereas it must be an integer if the corresponding item is an integer number or an alphanumeric word.

Consider the following seven data records to illustrate the asterisk, automatic-generation input features.

```
1 2 3 4 5 6 7 8 9 10 11 12 /  
2 1 *2 4 *=2 1 *=2+2 ** /  
*+1 0 *=3 0 *=5 ** /  
*+1 0 *=11 /  
*+1 *12 /  
*+1 ** /  
** /
```

Twelve data items are defined by each of these records. Each of the last six records is translated into the same internal record which is

```
2 1 3 4 4 4 4 1 3 5 11 12 /
```

If this particular sequence of data were required within a data deck, the last five records could be replaced by the single record

```
*+5 ** /
```

to achieve the same results.

Generation of alphanumeric items is illustrated by the following two records.

```
TY1 TO TY15 *=2+2 10.0 /  
*+2 1 0 2 3 4 1.0 /
```

These records are equivalent to the following records:

```
TY1 TO TY15 TY17 TY19 10.0 /  
TY2 TO TY17 TY20 TY23 11.0 /  
TY3 TO TY19 TY23 TY27 12.0 /
```

In some cases, the data generation capabilities can be used even though some of the generated items are not required. The "extra" data are simply ignored. If this situation is allowed, it is noted accordingly in the detailed record descriptions presented in the User's Manual.

5.1.3 Data Comments

User-defined comments, in the form of text, may be embedded in a data deck for conveniences of data identification. User comments, which do not affect any of the data preprocessing activities, can be included within a data deck in the following ways:

- a) Comments may be input at the end of a data record. The / symbol is input followed by the comments, which may continue to the right-hand boundary of the card.
- b) All characters in a record beginning with the two characters */ are treated as user comments. The first word in a record of this type must not be MODE1 or MODE2.
- c) Comments enclosed between the characters *(and) may be inserted any place within a data record. Pairs of parentheses, data-item delimiters and record termination characters may be included within this type of comment.

User comments of types (b) and (c) may continue onto multiple cards, as required. Continuation and termination of the data record are described in section 5.1.1 for the MODE1 and MODE2 input formats.

Examples: 10 2.0 3.0 100.0 / DEFINITION OF NODE 10
 */ NODE 11 IS DEFINED BY THE NEXT RECORD/
 11 *(THIS IS NODE 11) 4.0 3.0 100.0/

5.2 STRUCTURAL MODEL

The process of structural modeling involves assembling an aggregate of node points and finite elements. The geometric and material properties, plus the mass density of the structural finite elements are defined by the STIFFNESS data set. The nodal grid is defined to conform to the structural arrangement, and the element grid is defined to represent equivalent structural characteristics of the real structure. These data, together with boundary conditions, define the structural model for static and dynamic analyses. The mass associated with the structural model can be calculated directly from the structural model data. Additional mass distribution capabilities, such as finite-element modeling of distributed nonstructural masses, are discussed in section 5.3. If only a weights analysis is to be performed, stiffness (structural) finite elements are not required. In this case, a mass model is used that is comprised of a nodal grid, boundary conditions, and mass data (ref. sec. 5.3).

A structural model is identified by an integer SET number in the following required input data sets.

- a) NODAL
- b) BC (Boundary Condition)
- c) STIFFNESS

As many as 36 different structural models (SET numbers 1 through 36) can be defined in one ATLAS job. Corresponding to each structural model, 10 different groups of boundary conditions can be specified. Each group of BC data is identified by an integer STAGE number. Therefore, a structural model with a particular set of boundary conditions is identified by a SET number and a STAGE number.

The various components of a structural model are discussed in the following sections.

<u>Section</u>	<u>Description</u>
5.2.1 --	Coordinate systems used for referencing input data and output data
5.2.2 --	Nodal data used to define the model geometry
5.2.3 --	Boundary conditions and kinematics associated with the stiffness equilibrium equations
5.2.4 --	Characteristics of the ATLAS stiffness finite elements
5.2.5 --	Fundamental modeling considerations
5.2.6 --	Model data and solution-accuracy quality checks

Formatted printout of the structural model data (nodes, coordinate systems, elements and boundary conditions) can be generated by including appropriate "PRINT INPUT" statements in the execution-control deck (see sec. 6 and the User's Manual; secs. 206, 246, and 252). Pictorial plots of the node and element grids, as well as contour plots of the element properties can also be created using the GRAPHICS postprocessor (see sec. 6.3).

5.2.1 Reference Frames (User's Manual; sec. 146)

There is one GLOBAL (common) coordinate system used for referencing the input data and output data associated with a structural and/or mass model. This coordinate system is a right-handed, Cartesian triad denoted by X-Y-Z. In addition to the GLOBAL reference frame, the analyst may find it convenient to

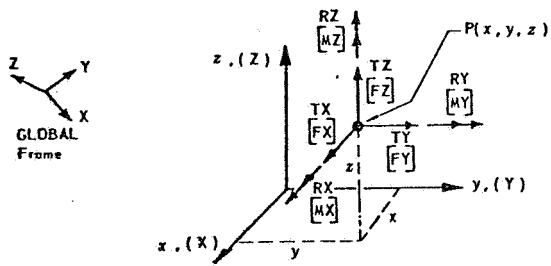
define additional local reference frames by the NODAL data set (User's Manual; sec. 146) for the following purposes:

- a) Node Definitions -- Input nodal data relative to local frames
- b) Node Kinematics -- Measure node kinematics relative to local analysis frames. Kinematics include the translation and rotation components resulting from static loads, the mode shape components associated with buckling or vibration analyses, and the boundary conditions (e.g., zero and nonzero support displacements).
- c) Load Directions -- Input magnitudes of mechanical forces relative to local frames at nodes
- d) Mass Inertias -- Measure input and output, concentrated-mass moments of inertia relative to local frames
- e) Geometry Components -- Define the geometry of the structural and/or mass model using one or more geometry components. Node locations for a component are input relative to a local frame. Orientation of a component relative to the total model is established by the origin and orientation of the local frame.

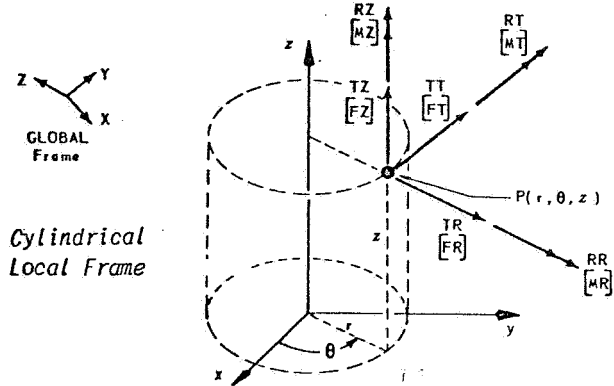
A local frame can be a rectangular x-y-z, a cylindrical r- θ -z, or a spherical ρ - θ - ϕ coordinate system. All coordinate systems are orthogonal and right-handed as shown in figure 5-3. The orientations of all local frames relative to the GLOBAL frame are designated by the analyst via the NODAL data set.

An input (definition) reference frame and an analysis (kinematic) reference frame are associated with each node. The analysis frame used to measure the kinematics of a node can be different from the input reference frame used to locate the node. Furthermore, the analysis frame of one node can be different from the analysis frame of another node. If local reference frames are not used, the input and analysis frame for all nodes is the GLOBAL frame.

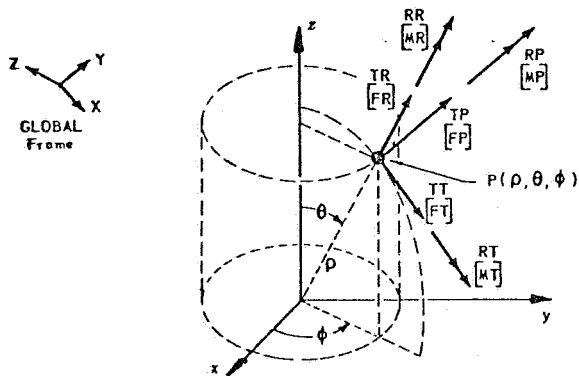
There is a set of six, ordered degrees of freedom associated with each node. The labels used to identify these kinematic freedoms and the labels used to identify the corresponding nodal forces, for each type of reference frame, are shown in figure 5-3.



Rectangular
Local (or GLOBAL)
Frame



Cylindrical
Local Frame



Spherical
Local Frame

NODAL FREEDOM NUMBER	NODAL REFERENCE FRAME TYPE					
	RECTangular		CYLindrical		SPHERical	
	Kinematic Label	Force Label	Kinematic Label	Force Label	Kinematic Label	Force Label
1	TX	FX	TR	FR	TR	FR
2	TY	FY	TT	FT	TT	FT
3	TZ	FZ	TZ	FZ	TP	FP
4	RX	MX	RR	MR	RR	MR
5	RY	MY	RT	MT	RT	MT
6	RZ	MZ	RZ	MZ	RP	MP

Figure 5-3. Kinematic and [Force] Labels and Conventions

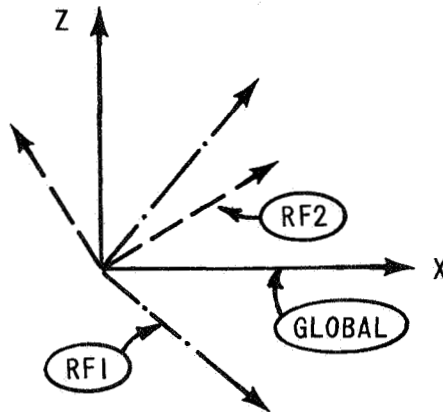
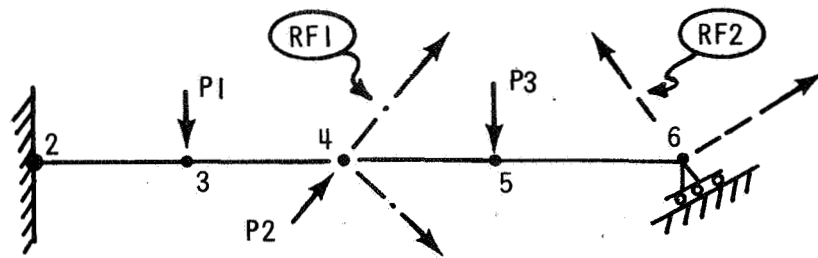
Figure 5-4 illustrates the use of several reference frames to define and analyze a simple structural model. Local frames, identified by the analyst as RF1 and RF2, are used to define a load direction at node 4 and an analysis frame at node 6, respectively. Use of the analysis frame RF2, for example, allows for convenient handling of the kinematics associated with the inclined support.

An "element reference frame" is associated with each of the structural finite elements and some of the mass finite elements. The sign conventions used for the input section properties and the calculated stresses are based on the orientation of these reference frames. Optionally, distributed element loads can be defined relative to the element reference frames. Each element reference frame is a right-handed, rectangular triad that is oriented according to a sequence of nodes. The nodes are designated by the analyst when the corresponding finite element is defined. Descriptions of all element reference frames are presented in appendices B and C of the User's Manual.

5.2.2 Geometry and Nodes (User's Manual; secs. 126, 146, and 246)

The geometry of the physical structure is idealized as a network of nodes (points) in three-dimensional space. Structural characteristics are idealized by finite elements that are interconnected at the nodes. It is not necessary, however, to have each node connected to an element. Auxiliary nodes, for example, can be used for orienting the principal bending axes of a BEAM element or for defining the natural material axes for orthotropic elements. A maximum of 4095 nodes can be used to define a total model or a substructure component.

A node can be defined with three or four coordinates. Nodes with four coordinates are referenced as mid-surface nodes used primarily for defining SPAR, COVER and CCOVER elements (see sec. 5.2.4). Coordinates of a mid-surface node defined relative to a rectangular reference frame, for example, are specified by a list of x, y, z and Δz components. The Δz component establishes two points located at $(z + \Delta z)$ and $(z - \Delta z)$ along a line parallel to the input z -axis. Thus, three points on a straight line are identified by a mid-surface node wherein the middle point lies on the "mid-surface" of the finite element. This concept allows for convenient definition of spar, rib, and stringer finite element depths. The input coordinates associated with the different types of frames are shown in figure 5-5.



REFERENCE FRAME	PURPOSE
GLOBAL	Define all nodes; Analysis frame for nodes 2,3,4 and 5; Define directions of loads P1 and P3.
RF1	Define direction of load P2
RF2	Analysis frame for node 6

Figure 5-4. Example Use of Local Reference Frames

Type of Nodal Reference Frame	Input Coordinates			
	c1	c2	c3	c4
GLOBAL	X	Y	Z	ΔZ
REC	x	y	z	Δz
CYL	r	θ	z	Δr
SPH	ρ	θ	ϕ	$\Delta \rho$

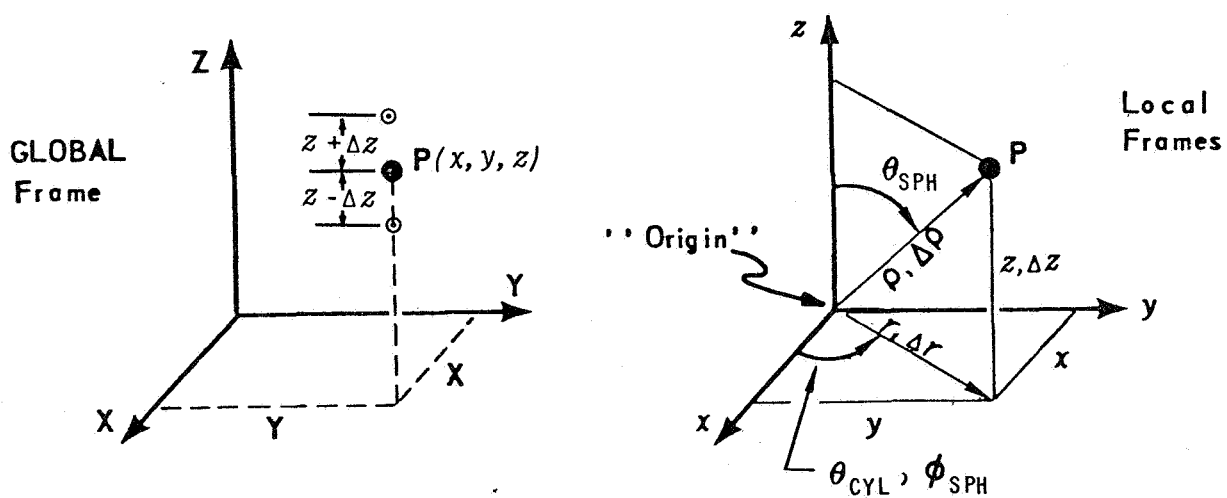


Figure 5-5. Nodal Reference Frame Coordinates

Each node is identified by a different integer number. Node numbers and coordinates can be input explicitly, generated from a sequence of equally-spaced or variably-spaced nodes, or they can be based on the exterior geometry of a component of the model. Each of these options, as described below, can be used to create a NODAL data set.

Explicit definition of node number 24, for example, is input by a data record with the following form:

```
24  6.0  35.0  2.0  /
```

Generation of the string of nodes 24,26,28,30,32,34, spaced according to the proportionate distances 1.0,0.5,0.5,.25,.25 between the pairs of nodes, can be designated by the following data record.

```
24  6.0  35.0  2.0  TO  34  8.0  49.0  3.2  
    BY 2 OF 1.0  0.5  0.5  .25  .25  /
```

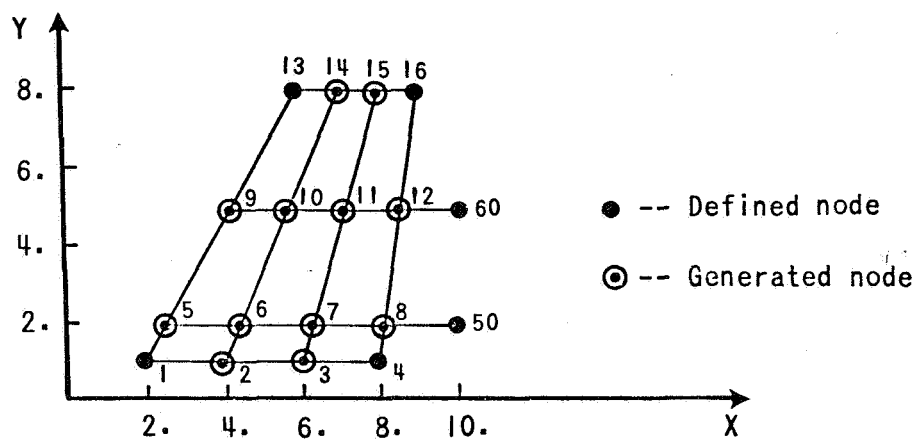
Coordinates of the generated nodes are obtained by linear interpolation of the specified coordinates for nodes 24 and 34.

Additional examples of nodal data records are shown in figure 5-6. In this case, a grid of 16 nodes is defined by four data records. Note that previously-defined nodes can be referenced without their coordinates in node generation records (e.g., 1 and 13 in the third record of fig. 5-6).

Other ATLAS capabilities allow the three-dimensional surface geometry of one or more components of a structural model to be defined (controlled) by one or both of the following:

- a) ATLAS GEOMETRY data sets
- b) Geometry data prepared for the NASA-LRCGEOM (NASA/Langley aerodynamic configuration) programs

Automated interfaces allow NASA-LRCGEOM data to be used in defining the geometry of ATLAS structural models. In both cases, surfaces are generated from a minimal amount of user-supplied data by numerical curve-fitting techniques and by enrichment of the geometry input data.



```

1  2.  1.  0.  TO  4  8.  1.  0.  /  EQUALLY-SPACED
13 6.  8.  0.  TO 16 9.  8.  0.  /  EQUALLY-SPACED
1  TO 13  BY  4  OF  1.  3.  3.  /  VARIABLY-SPACED
**+3  1  0  1  ** /  VARIABLY-SPACED
50 10.  2.  0.  /  EXPLICIT
60 10.  5.  0.  /  EXPLICIT

```

Figure 5-6. Example Nodal Data

An ATLAS geometry component is defined by user-selected longitudinal and cross-section control curves that establish its surface. Typical geometry components are shown in figure 5-7. The geometry data are defined relative to a local rectangular reference frame. The origin and orientation of the input reference frame establish the orientation of a component relative to the total model. Coordinates for surface nodes and mid-surface nodes can be extracted from the geometry component data in the following ways:

- a) Definition of nodes by direct input of only 2 coordinates per node
- b) Definition of nodes located by the longitudinal control curves of a component
- c) Generation of a sequence of equally-spaced nodes located relative to the pre-defined surface geometries
- d) Generation of a sequence of variably-spaced nodes located relative to the pre-defined surface geometries

Cutting planes positioned by the user to extract coordinates for nodes in the planes need not be restricted to the locations used in defining the cross-section control curves.

5.2.3 Boundary Conditions (User's Manual; secs. 106 and 206)

As shown previously in figure 5-3, the kinematics (response) of each node are described by six degrees of freedom; three translations and three rotations measured relative to the selected node analysis reference frame. The six degrees of freedom of a mid-surface node are defined at its mid-surface point. Motions of the two points located by a mid-surface node "delta" coordinate are defined by rigid transformations of the mid-surface point kinematics. Overall structural response is described by the aggregate nodal kinematics.

Boundary condition (BC) input data allow one of the following activities to be imposed on each nodal degree of freedom:

- a) FREE -- Unconstrained freedoms that are reduced from the gross (overall) matrices during generation of reduced matrices

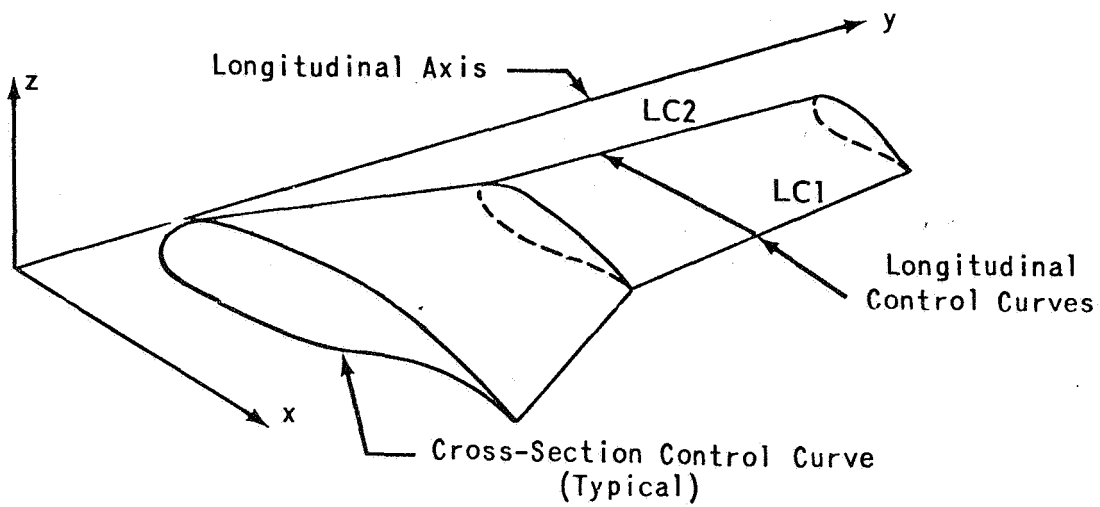
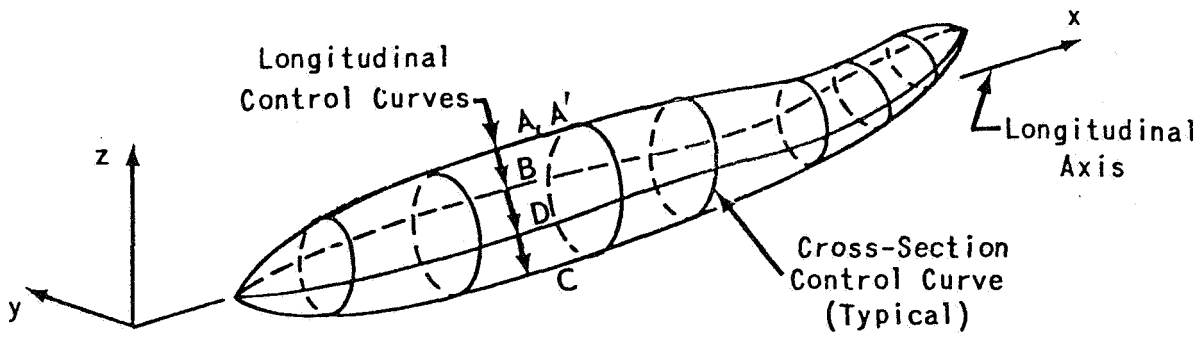


Figure 5-7. Typical Geometry Components

- b) RETAIN -- Unconstrained freedoms that are retained in the reduced matrices generated during some solution processes. Those freedoms to be used explicitly in the equations for a vibration analysis or a buckling analysis must be retained. Ordering of the retained freedoms in a particular sequence, a BC data option, allows for convenient use and interpretation of the calculated mode shapes. For a substructured analysis, a freedom of one substructure that interacts (joins structurally and kinematically) with a corresponding freedom in another substructure must be retained.
- c) SUPPORT -- Freedoms for which displacements are specified. A supported freedom is either rigidly fixed (not allowed any displacement), or it is displaced as specified. Nonzero support displacements can be input partially or wholly either within the BC data set or within the LOADS data set. For a stress analysis, a sufficient number of freedoms must be supported to satisfy overall equilibrium and internal equilibrium of the structure. A reaction force is calculated for each supported freedom.

The last activity specified for a particular freedom governs. It is sometimes convenient, for example, to RETAIN or SUPPORT a large group of nodal freedoms using a data-generation capability, and then FREE selected freedoms within the group. If no activity is specified for a freedom, it is assumed to be a FREE freedom. All inactive freedoms (those with zero stiffness) are automatically ignored during the solution process.

The governing system of stiffness equilibrium equations can be written as follows:

$$\begin{bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{bmatrix} \begin{Bmatrix} D_1 \\ D_2 \\ D_3 \end{Bmatrix} = \begin{Bmatrix} L_1 \\ L_2 \\ L_3 \end{Bmatrix} + \begin{Bmatrix} 0 \\ 0 \\ R \end{Bmatrix}$$

where the kinematic freedom types are identified as

- | | | |
|---|---|---------|
| 1 | = | FREE |
| 2 | = | RETAIN |
| 3 | = | SUPPORT |

The gross matrices in the foregoing equation are partitioned as follows:

K_{ij} = Stiffness matrices generated from the finite elements used to model the elastic structural characteristics

- Di = Displacement matrices. Matrices D1 and D2 are calculated, whereas matrix D3 is created from the user-specified SUPPORT displacements.
- Li = Thermal and mechanically-applied load matrices created from the LOADS input data
- R = Reaction force matrix calculated for the supported freedoms

Each column of the Di, Li and R matrices corresponds to a particular loadcase.

A maximum of 10 different groups of boundary conditions can be specified for a structural model. Each group is assigned a different STAGE number to identify different arrangements of the freedoms within the FREE, RETAIN and SUPPORT matrix partitions.

Elastic structural supports can be idealized using the ATLAS SCALAR spring elements (see sec. 5.2.4). This element constrains the motion of a node relative to ground. The structural equilibrium requirements for a stress analysis, for example, can be satisfied partially or wholly by use of SCALAR elements and supported freedoms.

When a structure and its loading have one or more planes of symmetry, the size of the resulting data matrices can be reduced significantly. Only half of the structure need be idealized and analyzed if there is one plane of symmetry, one fourth for two planes, and one eighth if three planes of symmetry are considered. Using this procedure requires symmetric and antisymmetric displacement configurations to be identified relative to each plane of symmetry. By supporting (rigidly fixing) appropriate freedoms in the planes of symmetry, symmetric and antisymmetric sets of displacement configurations are imposed.

For symmetric displacements, any movement of a point on one side of the plane of symmetry is accompanied by a corresponding mirror-image movement of the reflected point as shown in figure 5-8. Symmetric motion of all such points relative to the plane of symmetry is enforced by prohibiting the points in the plane from translating out of the plane or rotating about any axis lying in the plane. Therefore, antisymmetric displacements are constrained by using a data record of the following form:

```
SUPPORT ASYM IN SURFACE 2 /
*/ THE 1-3 PLANE IS A PLANE OF DISPLACEMENT SYMMETRY. THAT IS,
*/ SYMMETRIC MOTION RELATIVE TO THE 1-3 PLANE IS ALLOWED /
```

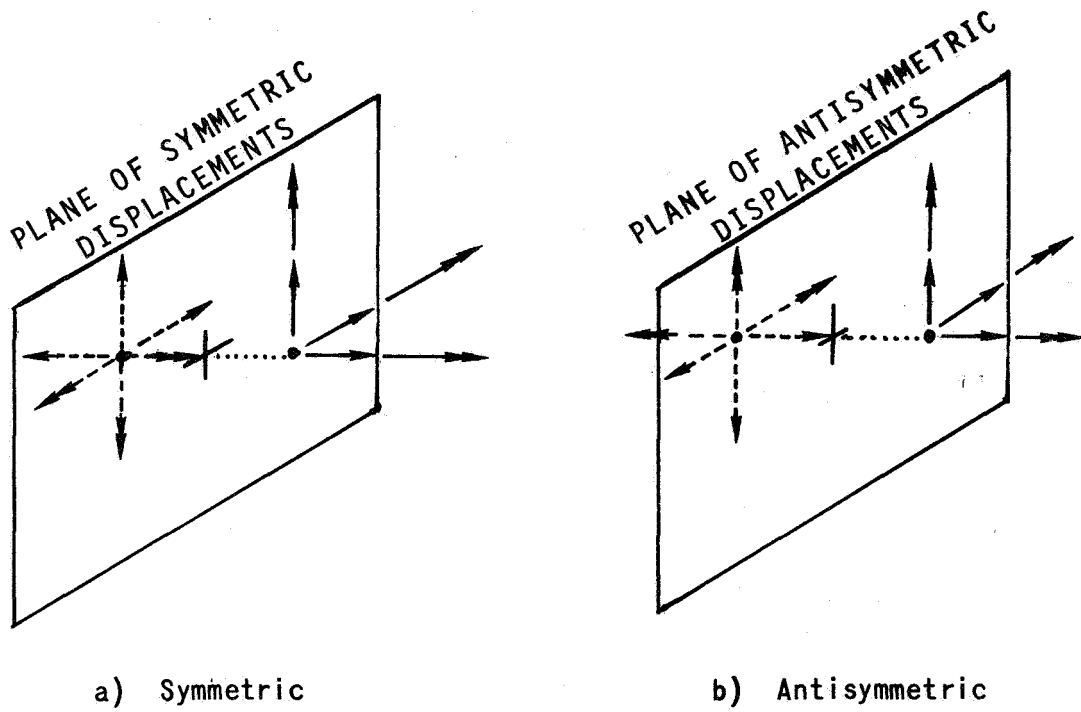


Figure 5-8. Symmetric and Antisymmetric Displacements

For antisymmetric behavior, the necessary supports in the plane of symmetry are just the counterpart of those used for symmetric motion. Thus, translation within the plane and rotation about an axis normal to the plane must be prohibited. The corresponding data record has the following form:

```
SUPPORT SYMM IN SURFACE 2 /  
*/ THE 1-3 PLANE IS A PLANE OF DISPLACEMENT ANTISYMMETRY.  THUS,  
*/ ANTISYMMETRIC MOTION RELATIVE TO THE 1-3 PLANE IS ALLOWED /
```

Symmetric and antisymmetric constraints are applicable to the static and dynamic linear analysis of all structures with symmetry characteristics. If a structure with one plane of symmetry is subjected to an unsymmetric load, a complete analysis is performed by decomposing the load into its symmetric and antisymmetric components. Half of the structure is analyzed with the two sets of boundary conditions and corresponding load components. The displacement response for the total model is then obtained by taking the sum of the displacements resulting from the two analyses.

Structures that are axisymmetric (bodies of revolution) or have cyclical symmetry characteristics can also be analyzed by using the foregoing procedure. In these cases, the surfaces of symmetry are defined by using cylindrical or spherical node-analysis reference frames.

Generally, the actual support conditions for a structure provide sufficient kinematic constraints so that rigid body motions are not allowed. If this is not the case when a stress analysis is to be performed, a sufficient number of supported freedoms must be selected to satisfy overall equilibrium. Additionally, any mechanisms internal to the structural model (rigid body motion of certain parts of the structure) are not allowed for a statically loaded model. Mechanisms are eliminated either by using additional elements to provide additional node connectivity or by supporting additional nodal displacements. Any number of rigid body modes can be inherent to the structural model when vibration analyses are performed.

A displacement/stress analysis of a physically unsupported structure subjected to a self-equilibrated system of loads is performed by selecting an artificial set of statically determinate freedoms to be supported for the analysis. Provided the applied loads are accurately balanced, the reaction forces corresponding to the supported freedoms should be zero.

Nonzero support displacements are used typically to represent settlement of the physical structural supports or to impose displacement boundary conditions when performing a detailed

stress analysis of a portion of the total structural model. The latter situation occurs when the stress distributions in certain regions of a complex structure do not provide satisfactory accuracy. In this case, a refined element grid is defined for the local area of concern, specified displacements are applied at the boundary of the local region, and a separate analysis is performed. The specified displacements are those determined from a previous analysis of the total structure.

5.2.4 Stiffness Finite Elements (User's Manual; secs. 152, 252 and appendix B)

Structural characteristics are idealized by stiffness (structural) finite elements that are interconnected at the nodes. A maximum of 32 767 stiffness elements can be used to define a total model (a SET) or a substructure component (see sec. 5.9) by a STIFFNESS data set.

Each element-definition data record contains the following sequence of information:

- a) Eltype -- Element type name or identifier (e.g., ROD, BEAM, or PLATE)
- b) Mcode -- Reference code for the material property table (e.g., M5, M52)
- c) Tcode -- Reference code for element temperature to be used when extracting material property values (e.g., T100, T+80, T-20)
- d) Userid -- User selected element number preceded by the letter N (e.g., N12, N530, N32500)
- e) Nodes -- List of node numbers that define the element
- f) Plist -- Either a property reference code (e.g., P2, P43, P90), or a list of element cross-sectional property values (e.g., 0.5 for a half-inch thick plate element)

If Mcode, Tcode, or Userid are not input, appropriate default values are used automatically. For example, if a number is not assigned to an element, it is automatically labeled by its input sequence number. The remaining data, (Eltype, Nodes and Plist) are required. Definition of an axial ROD element with a constant cross-sectional area of 0.75 that connects nodes 12 and 8 can be input by the following data record:

```
ROD 12 8 0.75 /
```

Use of reference codes helps to minimize the amount of required input data. Material tables, as defined by a MATERIAL data set (User's Manual; sec. 140) include the density, elastic constants, and allowable stresses. Isotropic, orthotropic, and composite material property tables are identified by reference codes. A number of standard (commonly used) isotropic material table codes are provided for direct referencing.

Reference codes for different lists of element cross-sectional property values are defined as part of the STIFFNESS data set. This feature is particularly useful when the same list of properties (as many as 13 values) applies to many of the finite elements.

Options are provided for generating sequences of elements provided they are of the same type, have the same temperature, and the same material and section property values. A grid of ROD elements that connect all the nodes shown in figure 5-4, for example, can be created by the following data records (a cross-sectional area of 1.0 is assumed).

```
ROD  1  2  1.0  TO  3  4  /  
*+3  0  4  4  0.  0  4  4  /  
ROD  8  50  1.0  /  
ROD 12 60  1.0  /
```

The different types of stiffness elements provided by ATLAS for structural modeling can be grouped according to their geometry.

- a) Scalar Element
- b) Rod and Beam Elements (One-Dimensional)
- c) Plate Elements (Two-Dimensional)
- d) Solid Elements (Three-Dimensional)
- e) Built-Up Elements (Combinations of the One- and Two-Dimensional Elements)

The geometry and the number of nodes used to define each element type are summarized in table 5-2, whereas table 5-3 summarizes the following element characteristics:

- a) Properties -- Material and geometric
- b) Stiffness -- Load-carrying stiffnesses relative to the element reference frame plus geometric (differential) stiffness capability
- c) Loads -- Types of loading applicable to the element
- d) Mass -- Type of elemental mass distribution

Table 5-2. The ATLAS Stiffness Elements

ELEMENT		NUMBER OF NODES (1)		GEOMETRIC SHAPE (2)
TYPE	NAME	STRUCTURAL	AUXILIARY	
Scalar	SCALAR	1	(2)	
Rod/Beam	ROD	2	0	
	BEAM	2	(1)	
		2	(2),(3)	
	SROD	3	0	
Plate	PLATE and CPLATE (Membrane)	3	0	
		3	3	
		4	0	
		4	4	
	GPLATE (Membrane and/or bending)	3	0	
		3	3	
		4	0	
		4	1	
		4	4	
		4	5	
ELEMENT		NUMBER OF NODES (1)		GEOMETRIC SHAPE (2)
TYPE	NAME	STRUCTURAL	AUXILIARY	
Plate (Cont'd)	SPLATE (Shear only)	4	0	
		4	4	
Solid	BRICK	8	(3)	
		20	(3)	
		32	(3)	
		44	(3)	
Built-up	SPAR	2 (Mid-surface nodes)	0	
		3 (Mid-surface nodes)	0	
	COVER and COVER	4 (Mid-surface nodes)	0	

(1) Auxiliary nodes denoted by (N) are optionally used for orientation of the element or its material axes.

(2) Rigid element components (offsets and rigid connections) are illustrated by

① Auxiliary nodes denoted by (N) are optionally used for orientation of the element or its material axes.

② Rigid element components (offsets and rigid connections) are illustrated by

Table 5-3. Characteristics of the ATLAS Stiffness Elements

ELEMENT NAME	PROPERTIES							STIFFNESS					LOADS			MASS	
	Orthotropic Material	Composite Material	Orthotropic Smeared Stiffness	Element Offset	Warped Element	Variable Cross Section	Axial	Torsional	Shear	Moment	Membrane	Geometric	Distributed	Thermal	Inertia	Lumped	Consistent
SCALAR							●	●	●	●							
ROD						●	●				●	●	●	●	●		
BEAM				●		●	●	●	●	●		●	●	●	●	●	
SROD						●	●					●	●	●	●		
PLATE	●		●	●	●				●		●	●	●	●	●	●	
CPLATE		●		●	●				●		●		●	●	●	●	
GPLATE	●			●	●	●			●	●	●	●	●	●	●	●	
SPLATE					●				●			●		●	●		
BRICK	●					●	●		●		●	●	●	●		●	
SPAR						●	●		●	●	●		●	●	●	●	
COVER	●		●		●		●		●	●	●		●	●	●	●	
CCOVER		●			●		●		●	●	●		●	●	●	●	

The characteristics and application of each element type, and the specific elements available within each group are described in the following sections. Detailed descriptions of the ATLAS stiffness elements, including the element reference frames, section properties and materials, element loading capabilities, and element stress definitions, are presented in appendix B of the User's Manual.

5.2.4.1 SCALAR Element

Use of this element allows one to six "elastic support" stiffnesses (3 translational and 3 rotational) to be specified for a node. The element connects the selected node to ground by the specified "spring" stiffnesses that act in directions which are established by an orientation node. The resulting nodal forces and moments acting on the element from ground are displayed in the stress output.

5.2.4.2 Rod and Beam Elements (One-Dimensional)

A one-dimensional element is assumed to be slender enough so that its behavior is described in terms of properties associated with a single straight line representing its elastic axis. One-dimensional elements are used to model beams, columns, frame members, truss members, taut cables, torsion bars, stiffeners and flanges in semimonocoque structures, elements of rings and arches, etc. Curved members can be modeled by using an adequate number of straight elements.

ROD

This element provides axial stiffness only. Derivation of the stiffness is based on the assumptions of constant axial load and a uniform cross-sectional stress distribution. A linear area variation between its two end nodes is allowed. Stress output consists of the axial force and axial stress at the end nodes.

BEAM

This element resists axial extension, torsion, shear, and bending about each of two principal axes in its most general form. Various combinations of zero and nonzero values of the cross section properties, which can vary linearly between its two end nodes, can be used. It is assumed that the shear center axis and the elastic axis coincide. An orientation node can be used for establishing the directions of the principal bending axes. The elastic axis may be offset from one or both of the structural nodes. The end of an offset BEAM is treated as if it were connected to the structural node by a rigid extension. This capability is useful in modeling stiffened plates and shells or

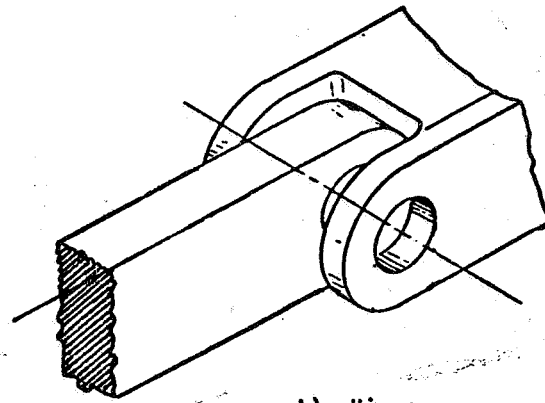
similar structures where frames and stiffening members have their neutral axes offset from a convenient reference surface, such as the skin surface in a semimonocoque fuselage structure. Hinges, sockets, and slides (slots) can be defined relative to the element reference frame at the beam end nodes as illustrated in figure 5-9.

A common mistake that should be avoided is specifying too many releases of member forces in trying to model hinges, rollers, etc. Torsion about the longitudinal axis of a BEAM should never be released at both ends of the member. Improper specification of end-force releases can lead to erroneous results caused primarily by elemental or model instabilities. Figure 5-10 illustrates the use of end-force releases, element orientation, and data record formats for a BEAM element model.

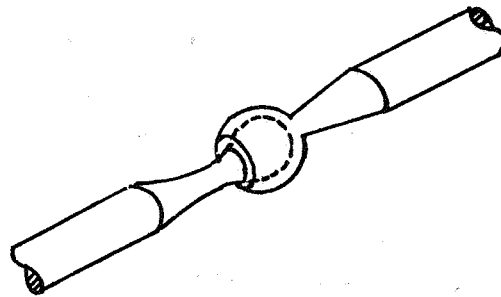
The lateral displacement function used in the stiffness matrix derivation is a cubic polynomial. This corresponds to a linear variation in bending moment and a constant shear which coincides with the exact solution for flexure of beams loaded at the end points (engineering beam theory). The effect of shear deformation on the transverse displacement can be included by specifying a nonzero value for the effective shear area. It is assumed that the shear is constant through the depth of the beam. For a typical I-section, the web carries essentially all of the shear load with a constant value of shear stress. For sections where the shear stress varies significantly through the depth, an "effective" shear area should be calculated by equating the strain energy of the "effective web" in a constant state of shear stress to the strain energy in the actual section with its true shear distribution for an arbitrary shear load. For a rectangular section, this results in an effective web area equal to 0.833 times the rectangular cross-section area. Caution should be taken to avoid inaccuracies when modeling shear areas for deep beams (which have a nonlinear distribution of axial stress) and for thin-walled open section members.

Torsional stiffness is treated according to St. Venant's theory (no warping) which assumes a constant torsional moment. The torsion constant, J , is the polar moment of inertia for solid shafts of circular cross section or for hollow shafts of annular cross section. For any other cross sectional shape, J is less than the polar moment of inertia of the section.

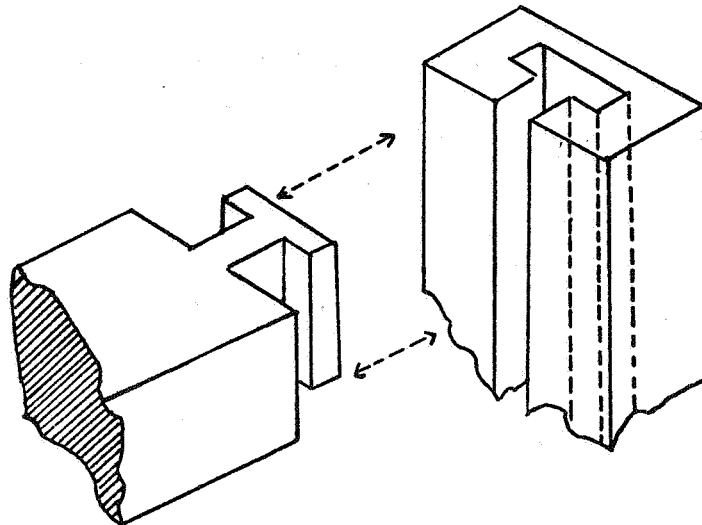
Stresses calculated for BEAM elements include the axial force, transverse shear forces, torque, and bending moments at the end nodes.



A) Hinge

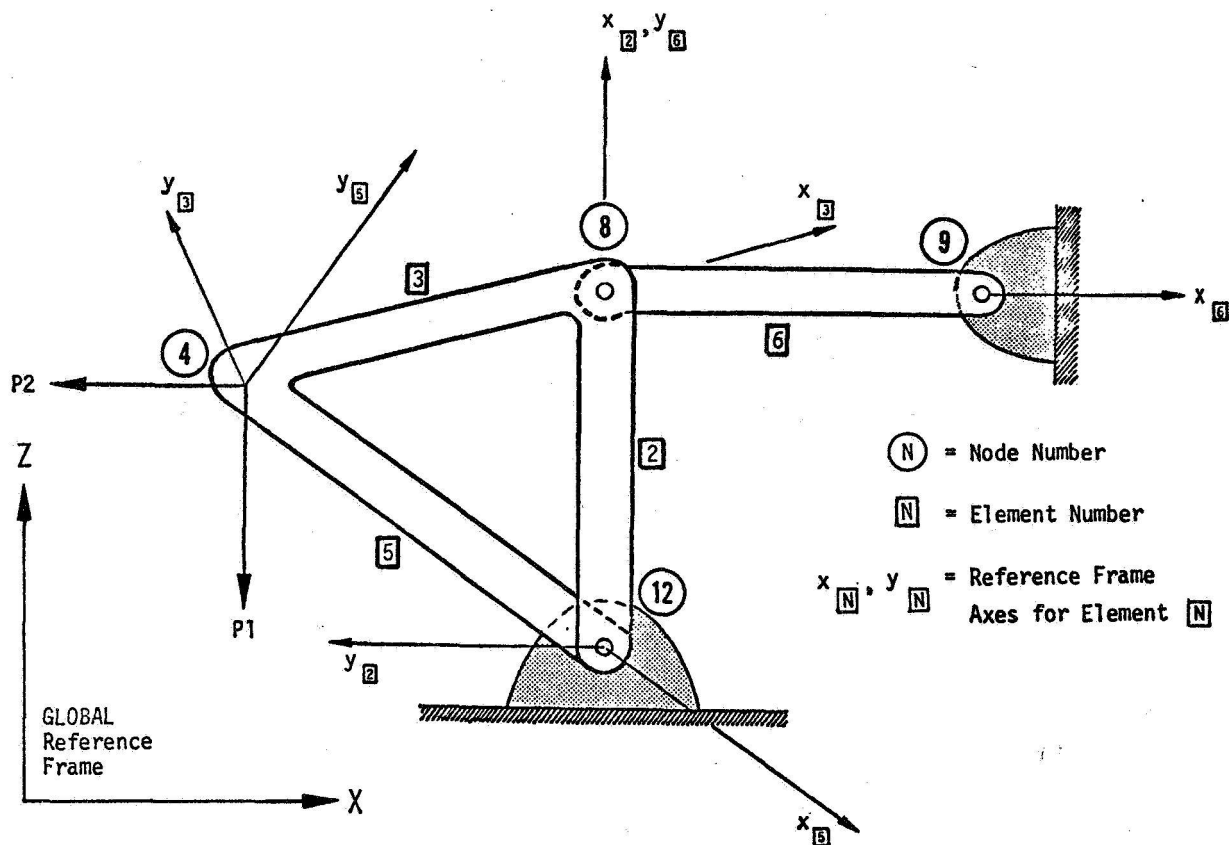


B) Ball and Socket



C) Slot

Figure 5-9. Typical End-Force Releases for BEAM Elements



(a) BEAM Element Model

BEAM N3	4	8	p1	p2	p3	
BEAM N5	4	12	p1	p2	p3	100000.
BEAM N2	12	8	4	p1	p2	p3 100.
BEAM N6	8	9	p1	p2	p3	100100.

(b) Example Data Records

NOTES:

1. Integer element numbers are assigned by the "Ni" data items
2. Section properties are input by the "pi" data items
3. The code 100000. denotes that the end moment M_z is to be released relative to the element reference frame at the second node used to define the element. That is, node 12 for element 5.
4. Node 4 is used to orient the x-y plane for element 2. An orientation node must be input for all BEAM elements that are parallel to the GLOBAL Z-axis.
5. The code 100. denotes that the end moment M_z at node 12 is to be released for element number 2.
6. The code 100100. denotes that the end moments M_z at nodes 8 and 9 are to be released for element number 6.

(c) Comments on the Data Records

Figure 5-10. Example Data Records Used to Define BEAM Elements

SROD

This element provides axial stiffness only, with the capability of transferring shear forces at an intermediate node to another compatible element (an eight-node SPLATE or another SROD). Derivation of the stiffness is based on the assumptions of a linearly varying axial load, uniform cross-sectional stress distribution, and the aforementioned shear transfer. A linear area variation between its two end nodes is allowed. Stress output consists of the shear flow along the element plus the axial forces and stresses at the end nodes.

5.2.4.3 Plate Elements (Two-Dimensional)

Two-dimensional elements are used to model "surfaces" as opposed to "line" structures as discussed in section 5.2.4.2. Although the ATLAS plate elements are planar, they are often used for analyzing thin shells by modeling the curved structure with a polyhedral surface. Membrane (in-plane shear and extension), bending (out-of-plane), and shear elastic behavior of orthotropic and composite materials can be investigated. In the derivation of the stiffness matrices for the family of ATLAS triangular and quadrilateral plate elements, it is assumed that there is no coupling between the membrane and bending behavior.

Generally, when plate elements are used to model a curved surface without stiffening members, as for a shell, both bending and membrane stiffnesses are used. To analyze a shell problem in which the response is primarily a function of membrane behavior alone, plate elements with only membrane stiffness are generally used. In this case, however, care must be taken to ensure transverse kinematic stability of the polyhedral assemblage by supporting (fixing) appropriate transverse-displacement degrees of freedom.

When using plate elements to represent bending behavior in a wing or box beam type of structure, the bending of the skin or cover plate about its own neutral axis is usually insignificant when compared with the overall bending stiffness of the section. In this case, membrane plate elements should be used since element bending stiffnesses would only complicate the problem unnecessarily, with the introduction of additional degrees of freedom. Typically, the surface elements are attached to ribs or spars in the transverse direction, and thereby are rendered kinematically stable. The lateral freedoms of those nodes on the surface, however, that are not attached to elements with transverse stiffness should be supported (held fixed) by input of corresponding boundary condition data. Supporting these freedoms will render the model kinematically stable and will not, otherwise, affect the quality of the results.

Except for the SPLATE element, the membrane surface (and neutral surface in bending) may be offset from one or more of the structural nodes used to define the ATLAS plate elements. An offset plate corner is treated as if it were connected to the structural node by a rigid extension. This capability is useful in modeling structural plates where the neutral surface is offset from a convenient reference surface.

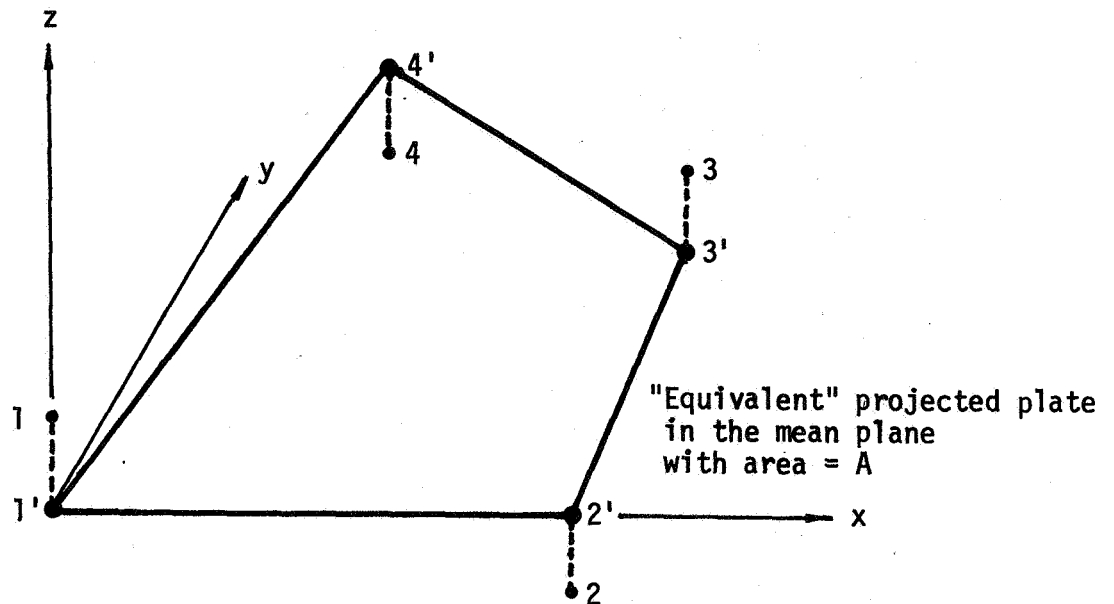
In the derivation of stiffness matrices for quadrilateral plates, it is generally assumed that the four corner nodes are coplanar. Since this is not generally true when the elements are placed on a curved surface, the corners are projected onto a mean plane that is defined by the vector cross-product of the diagonals, prior to calculation of the element stiffness matrix (see fig. 5-11). When the corners are not coplanar, overall equilibrium of the warped element is satisfied by automatic calculation of transverse "kick" equilibrium forces. Stresses are calculated for the "equivalent" plate, defined as the original element projected onto the mean plane. Experience has shown that a reasonable amount of warping can be tolerated without adversely affecting the results. When excessive warping occurs, an appropriate warning message is issued.

PLATE

This is a triangular or quadrilateral plate that resists membrane stiffness only. The element can have isotropic or orthotropic material, uniaxial smeared stiffening, and it can have offsets. A linear displacement field is assumed in the stiffness derivation for the basic triangular element. This corresponds to a constant stress/strain field over the triangular region. Four component triangles joined at a common internal vertex are used in the stiffness generation for a quadrilateral plate. The stiffness effects at the common point are reduced automatically so that the behavior of a quadrilateral plate is defined completely in terms of the corner nodes. When this type of element is used in areas of large stress gradients, a sufficient number of elements should be used so that the overall varying stress field is represented accurately enough by the constant stress/strain triangular regions.

The orthotropic smeared stiffening capability is convenient for modeling geometric orthotropy resulting from waffle construction, integral stiffening, corrugated sandwich construction, etc. Different effective thicknesses are input for stretching in each of two directions.

Stresses calculated for PLATE elements include the average values of the normal stresses and the shear stress, in addition to the axial stresses in the stiffeners.



1,2,3,4 = Specified element corner nodes; element diagonals are defined by 1-3 and 2-4

1',2',3',4' = Projected element corner nodes

x-y-z = Element reference frame for the equivalent plate; the mean plane is x-y.

A non-dimensional warping factor is calculated as $d/(A)^{1/2}$

where

d = Distance between the 1-3 and 2-4 diagonals

Figure 5-11. A Warped (Distorted) Quadrilateral Plate

CPLATE

This is a triangular or quadrilateral laminated plate that resists membrane loading only. The element can be comprised of up to ten orthotropic laminas, and it can have offsets. Each lamina is comprised of a number of layers of composite material oriented according to its fiber direction.

Derivation of the membrane stiffness for the basic triangular element is based on a constant stress/strain field as described previously for the PLATE element. The quadrilateral CPLATE element stiffness is derived in the same manner as discussed for the quadrilateral PLATE element.

The standard stress data calculated for CPLATE elements are the normal and shearing strains and stresses for the total element (multiple laminas). Optionally, the stress and/or strain components in each lamina can be calculated and displayed relative to the fiber direction of the lamina.

GPLATE

This is a triangular or quadrilateral plate with uncoupled membrane and bending stiffnesses based on the theory published in reference 5-1. Different thicknesses can be specified independently at the nodes for the membrane and bending characteristics. The element can have isotropic or orthotropic material, and it can have offsets. The stiffness matrix for the triangular element is based on the assumptions of constant membrane strain and linear curvatures for bending. Four component triangles joined at a common internal vertex are used in the stiffness generation for a quadrilateral plate. A linear membrane strain field and linear curvatures for bending are assumed in the stiffness derivation for these component triangles. The stiffness effects at the common point are reduced automatically so that the behavior of a quadrilateral plate is defined completely in terms of the corner nodes. No in-plane bending stiffness is provided by this element.

Stresses calculated for GPLATE elements include the average values of the normal and shearing stresses, in addition to the bending and twisting moments.

SPLATE

This element is a constant thickness, four- or eight-node quadrilateral shear panel that is based on Garvey's theory of uniform shear (ref. 5-2). With four nodes, shear forces are lumped at the corner nodes. With eight nodes, shear forces are

transferred at intermediate nodes on each edge of the plate to another compatible element (an SROD or another SPLATE).

Shear flows along the element edges, as well as the maximum shear stress are computed.

5.2.4.4 Solid Elements (Three-Dimensional)

Solid (brick) elements are used to model three-dimensional structural components that cannot be idealized adequately by an aggregate of "line" and/or "surface" elements. Typical use of solid elements include three-dimensional analysis of pressure vessels, thick-shell structures as encountered in pipe joints, propellers, blade components of engines, and landing gear components of aerospace vehicles.

BRICK

A family of isoparametric solid elements, each of which is defined by 8 corner nodes and 12 edges. The basic BRICK element is an 8-node hexahedron. One to three intermediate nodes can be specified on any of the 12 edges. Therefore, anywhere from 8 to 44 nodes are used to define each element. The geometry of an edge is described by a linear, second-order, third-order, or fourth-order polynomial depending on whether 0, 1, 2, or 3 intermediate nodes are used, respectively. The kinematics of each node are defined by three translational degrees of freedom. Displacement compatibility is automatically ensured at the common nodes of adjacent elements. The material can be isotropic or orthotropic with principal material directions identified by the user.

The elasticity equations of the three-dimensional continuum are formulated on the basis of three translational displacement components at each node. The 8-node BRICK, with a linear displacement field, defines a region of constant strain. As for the membrane PLATE element when used in two-dimensional areas with large stress gradients, a large number of 8-node BRICK elements must be used to represent accurately a varying stress field in three-dimensional regions. Alternatively, fewer numbers of the higher-order elements (20-node, 32-node or 44-node) are required to idealize regions with varying stress fields, stress concentration regions, and in the vicinity of concentrated loads. Experience has shown that excellent results can be obtained by using 32-node BRICK elements, which are based on a quadratic strain field approximation, in regions of this type. Another advantage of the higher-order elements is the degree to which the more complex structural boundaries with curvatures can be idealized.

Problems observed in the use of three-dimensional elements are the difficulty in which the finite-element model is created and visualized, and the large amount of data handling required for processing. As with any complex finite-element model, judicious use of the postprocessing capabilities, particularly graphical displays, alleviates considerably the problem of ensuring structural integrity. Three-dimensional analyses typically require a large number of nodes to predict reasonably accurate structural response. This results in a large number of simultaneous equations which may tax the data storage capacity of the computer, if a substructured model is not used. As such, the approach to problem solving should include decisions as to whether more complex elements with better accuracy should be used instead of the simpler finite elements with their limited accuracy.

The standard stress data calculated for BRICK elements are the values of the three axial stresses and the three shear stresses at the centroid of the element. Optionally, the same six stress components relative to each node of the element can be requested.

5.2.4.5 Built-Up Elements

Elements in this category are comprised of two or more of the basic one- and two-dimensional elements with specific assumptions as to how they are connected and allowed to deform. Since they are derived for modeling specific structural concepts, their general use is more restricted than the basic one-, two- and three-dimensional element types discussed previously. The built-up elements are more efficient, however, in their intended use.

The three built-up elements provided by ATLAS are identified as the SPAR, COVER, and CCOVER elements. Their primary use is for idealizing box beam type constructions and wing-like configurations. SPAR elements are used to model the ribs and spars, whereas the COVER and CCOVER elements are used to model the structural surfaces. The geometry of these elements is based on the definition of a "mid-surface" that is located such that the "delta" coordinate to the upper surface in one direction is the same as the "delta" coordinate to the lower surface in the opposite direction at a particular location on the mid-surface. The kinematic response of the structure is described in terms of the selected mid-surface points.

SPAR

This element is comprised of a quadrilateral pure-shear web of uniform thickness, an upper cap and a lower cap with axial stiffnesses that can have a linear area variation between the

ends, and two rigid posts at either end of the web that are parallel in the undeformed state. The element resists extension and bending moment by the caps, and transverse shear by the web.

In the derivation of the element stiffness, it is assumed that a uniform shear flow exists between the web and the caps. Each cap is composed of two components; a directly-defined flange area plus a fraction of the web area that is to be considered effective in carrying bending moments. The locations of the cap areas can be offset at either end within the plane of the element. This capability is useful in modeling caps that have their neutral axis offset appreciably from the structural surface.

Stresses calculated for SPAR elements include an equivalent shear flow and maximum shear stress for the web, as well as average values of the axial loads and stresses in the caps.

COVER

This element is comprised of two triangular or quadrilateral PLATE elements (see sec. 5.2.4.3) that are separated by rigid posts between the corresponding corner nodes. Five degrees of freedom (two bending and three translation) are provided relative to the element reference frame at each node. Either the upper plate or the lower plate can be absent from this element, thereby providing a convenient method for modeling cut-outs in the structural surface.

Stresses calculated for COVER elements include the average values of the normal stresses and shear stresses in the upper and lower plates, as well as the axial stresses in the plate stiffeners.

CCOVER

This element is comprised of two triangular or quadrilateral CPLATE elements (see sec. 5.2.4.3) that are separated by rigid posts between the corresponding corner nodes. Five degrees of freedom (two bending and three translational) are provided relative to the element reference frame at each node. As for the COVER element, either the upper plate or the lower plate can be absent.

The standard stress data calculated for CCOVER elements are the normal and shearing strains and stresses for each CPLATE (multiple laminas). Optionally, the stress and/or strain components in each lamina can be calculated and displayed relative to the fiber direction of the lamina.

5.2.5 Fundamental Modeling Considerations

A fundamental responsibility of an engineer performing a finite element analysis is the selection of the number, type, and arrangement of structural elements needed to provide an adequate idealized representation of the real structure. The structural model must be detailed enough to give accurate results but not so large as to become prohibitive in terms of setup time or execution cost. The principal factors involved in this responsibility are discussed in the following subsections. Other aspects of modeling such as boundary conditions, mass distribution, loads, and temperature distribution are discussed in other sections herein.

5.2.5.1 Node Distribution

The number and distribution of nodes required for a proper idealization are dictated by the geometry of the structure, the type of results desired, the nature of the expected response, and the types of elements used for the model. Generally, good stress results will require more nodes and elements (a fine grid) than are necessary for good displacement results. Conversely, a vibration analysis for the fundamental natural frequency requires fewer nodes and elements (a coarse grid) than are necessary for a good displacement analysis. When multidisciplined analyses of a structure are performed, use of the single finer-grid model, with appropriate sets of boundary conditions, is generally more cost effective than using different grids.

The nature of the expected response can dictate the number of nodes and elements required for accurate results. A finer element grid is necessary, for example, in areas of large stress gradients (stress concentrations) than in areas where stress levels are relatively constant.

The selected types of elements influence the node distribution by the nature of the admissible displacement states assumed in the derivations of the element stiffness matrices. A particular displacement state implies a certain state of strain. In a plane stress analysis, for example, use of linear strain elements (the GPLATE element) rather than the simpler constant strain elements (the PLATE element) requires fewer nodes and elements for an equivalent accuracy of calculated stresses.

5.2.5.2 Choice of Element Type

Selection of element types for the model is influenced primarily by the structural behavior and the geometry (configuration) of the structure. Factors such as the type of analysis (static, dynamic), the load environment (concentrated,

distributed, thermal), and the desired results (displacements, stresses, frequencies) are generally secondary in nature. The types of finite elements provided by ATLAS are grouped in section 5.2.4 according to their geometry, wherein the characteristics of each element type are discussed.

A variety of structural modeling capability is provided within each category of element types. For example, in the one-dimensional category (ROD and BEAM elements), different degrees of complexity can be defined as required for the particular circumstance. Various combinations of zero and nonzero values of cross section properties, offsets, and kinematic constraints for a BEAM element provide a large number of element sub-classes. A pure torsion rod, for example, requires that only a nonzero value is specified for the polar moment of inertia, whereas an axial member requires only the area property to be nonzero. A BEAM element with only the area property being nonzero is identical to a ROD element. A two-dimensional frame analysis using BEAM elements generally requires nonzero values only for the area and the out-of-plane bending moment of inertia. All degrees of freedom with zero stiffness are automatically excluded from the problem solution. The analyst should always strive to use the simplest element options that are consistent with the analysis requirements. Otherwise, unnecessary complications, increased size and cost of the analysis, reduced chances of a successful solution, or a reduced quality of the results are typically encountered.

In some cases, a "stick model" is used to idealize a structure by using only BEAM elements. Although a stick model appears to be very simple, considerable expertise is required to construct one that accurately represents a complex real structure. The simplified assumptions generally made to create a stick model can only yield overall structural deformation characteristics, identify areas where high stresses are likely to occur, and reveal general load paths within various parts of the structure. A stick model is generally used only to investigate how the structure reacts to loads and may not resemble the actual structural geometry. Occasionally, non-offset BEAM elements are used to model "rigid" links in a model. In this case, the stiffnesses of the artificial members should be approximately 10⁴ times larger than the connecting real-member stiffnesses so that ill conditioning of the stiffness matrix (and an inaccurate solution) is avoided.

5.2.5.3 Element Grid Layout

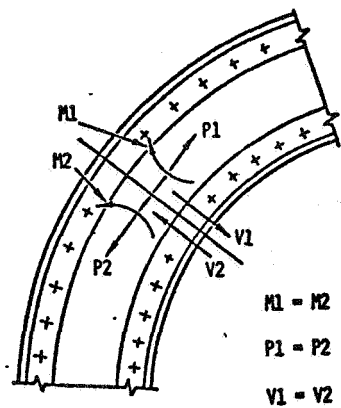
One of the first decisions that the analyst must make during the structural idealization process is the extent of detail to be included in the model. The general tendency is to include

details in the model that will not significantly affect the final results based on the problem assumptions. For the more complex models, there is a greater possibility of modeling errors, and may result in prohibitive solution costs and untimely solutions. If, however, a complex model is required, use of a substructured analysis should be considered; the extent of modeling detail should be checked carefully; and the "road maps" used for modeling, as well as the solution process, must be established carefully to allow flexibility in the management of the problem/solution data during the problem-solving.

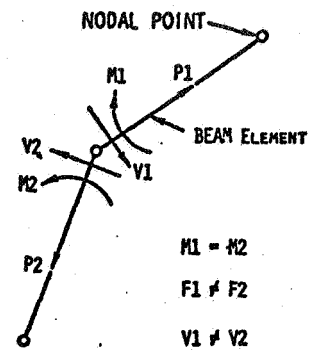
The arrangement of two- and three-dimensional elements may have considerable influence on the quality of results and their usefulness. Element stresses in particular, which are referenced to the local coordinate system for each element, are affected by the arrangement of the elements. A haphazard and irregular grid layout often gives erratic stress results that are difficult to interpret and are inconvenient to use. Figure 5-12 illustrates how the calculated element stresses must be interpreted to associate them with the real structure. Displacement data, which are referenced to the node analysis frames, are generally not as erratic in nature as stress data. These effects are a direct result of the displacement formulation of the finite element method.

Continuity of the displacement field is ensured within elements and at the nodes of adjacent elements, whereas continuity of the stress field between elements is not directly ensured. Interelement stress continuity is generally improved by using finer element grids and/or higher-order elements. Plate elements based on linear strain assumptions, for example, produce stress states that vary more uniformly than those calculated for plate elements based on constant strain (piecewise linear displacement) assumptions. Likewise, quadrilateral plate elements with width-to-length ratios (aspect ratios) of approximately one, in areas of stress gradients, give better results than those with long slender shapes (aspect ratios larger than 2).

When using different element types to construct a model, differences in the elemental degrees of freedom must be recognized. Use of BRICK elements, for example, intermixed with BEAM and GPLATE elements requires care during the structural modeling to account for proper transfer of bending moments between the different element types.

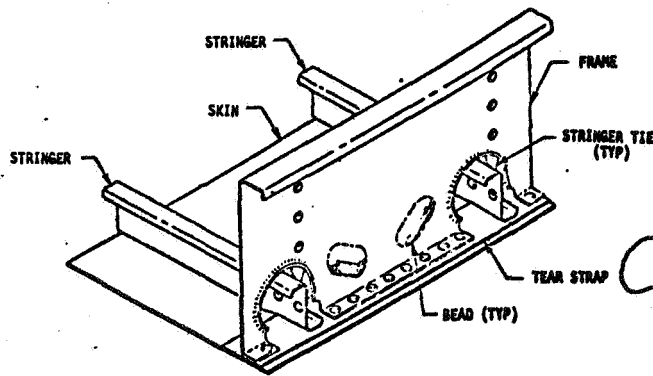


ACTUAL HARDWARE

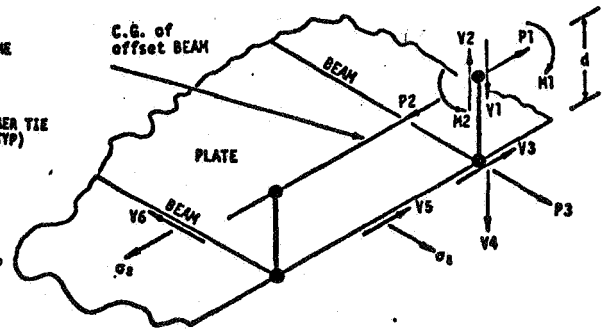


IDEALIZED MODEL

(a) Beam Model

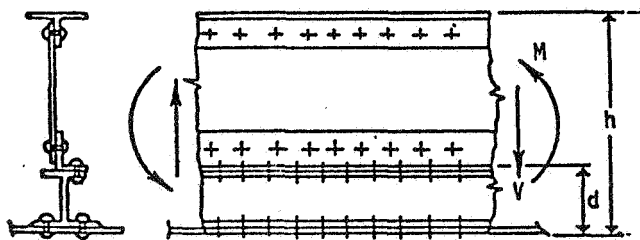


ACTUAL HARDWARE

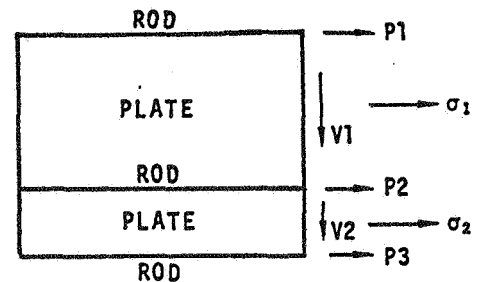


IDEALIZED MODEL

(b) Beam/Plate Model



ACTUAL HARDWARE



IDEALIZED MODEL

(c) Rod/Plate Model

Figure 5-12. Element Stress Interpretation

In general, a regular, orderly arrangement of elements should be used. A logical layout might follow any convenient natural coordinate system (e.g., rectangular or polar coordinates) that fits the structural geometry. This approach, however, is generally restricted to relatively simple geometric configurations with regular boundaries. One approach is to use a quadrilateral grid that approximates the anticipated stress trajectories (lines everywhere tangent to the directions of the principal stresses). The advantages of using this approach are as follows:

- a) The grid tends to fit the geometry of the structural boundary.
- b) Grid lines are closer together where stresses are larger in magnitude. Elements tend to be smaller in size in regions of stress concentration where greater accuracy is desired. Smooth transition from a fine grid (smaller elements) to a coarse grid (larger elements) is provided automatically.
- c) The calculated stresses are close to being the principal stresses.
- d) Oscillations of stresses in triangular elements fitted to this type of grid, particularly at structure boundaries, are minimized.

Smooth element-size transitions that are consistent with the overall structural requirements should be used so that smooth variations of the displacement and stress fields are provided.

No specific rules or formula can be stated for determining the degree of grid refinement necessary for a sufficiently accurate analysis in a given situation. The analyst must visualize the appropriate configuration of the displacement or stress field, establish the grid points at which results are desired, and define the element grid layout accordingly. Basically, idealization of a structural problem is dependent on the geometry of the structure, the loading environment, and the boundary conditions. A discontinuity in any of these will usually require a finer element mesh.

In the case of complex aerospace vehicle structures, it is usually impossible to define a single element grid layout that can be used to determine stress concentration factors and detailed local stress distributions with satisfactory accuracy. Generally, a gross analysis is performed first with the primary objective being the delineation of primary load paths and the associated stresses in the main structural members. The results

of the gross analysis, combined with engineering judgement and experience in structural analysis, are then used to perform separate local analyses of areas where a more refined grid is employed. When isolating the region to be treated in the local analysis using a finer grid, a sufficiently large enough area must be used so that any irregularities in the applied boundary conditions will be "damped out," according to St. Venant's principle, without disturbing the stress field in the area of interest.

Displacements or nodal loads, as calculated by the gross analysis, are used as boundary conditions on the nodes that define the junction of the local region with the rest of the structure. When applying load boundary conditions, it is usually necessary to specify some of the displacements in order to prevent rigid body motion. Any unbalance in the applied load system will be reacted by forces at the supported nodes. Therefore, care must be taken to balance the loads precisely. Even a relatively small unbalance can cause serious perturbations in the displacement and stress fields in certain situations. Because of these concerns, it is generally more convenient to use displacement boundary conditions when performing the detailed local analyses.

5.2.6 Solution Accuracy Checks

The structural model input data are transformed by ATLAS into a system of simultaneous equations which represent a state of structural equilibrium. In solving these equations, errors in the structural model can be detected. The primary reasons for errors in the mathematical model, and the structural model quality checks provided by ATLAS during the problem solution process are described in the following. Other ATLAS solution quality checks pertaining to specific types of analysis are described elsewhere in section 5.

Unique solutions to a system of non-homogeneous simultaneous equations can be generated only if the equations are linearly independent. When two or more of the equations are exactly dependent, the coefficient matrix for the system of equations is singular. In this case, the mathematical model must be modified to remove the equation dependency so that a nontrivial solution exists. Thus, the model-definition data must be corrected.

More frequently, the situation arises when two or more of the equations are almost linearly dependent (ill-conditioned). A unique solution to such a system of equations requires the analytical calculations to be performed with a high degree of accuracy (a large number of significant digits must be used during computation). Any solution that requires more than 13

significant digits must be considered erroneous since the precision of a decimal number stored by the CDC 6600/CYBER computer is approximately 14 digits of accuracy. Obviously, the difference between a system of equations that requires 14 digits of accuracy (a very ill-conditioned system) and a singular condition is purely numerical.

In ATLAS, the Cholesky square root method is used to solve the structural equations of equilibrium. The structural stiffness characteristics are used to define the coefficient matrix for this system of equations. The first step in the solution process is to decompose the coefficient matrix into the product of a lower-triangular matrix post-multiplied by its transpose. In addition to performing the decomposition, the CHOLESKY processor calculates the number of significant digits lost (the pivot decay) during the computations. The number of significant digits lost during solution increases as the degree of equation dependency increases. All singularities and large pivot decays encountered during the matrix decomposition are identified in the printout, along with the corresponding kinematic freedoms. Loss of a large number of significant digits is generally corrected by modifying the boundary conditions and/or the element properties associated with the corresponding kinematic freedoms.

Experience has shown that up to 3 digits can be lost during the solution of well conditioned equations for smaller models, and up to 6 digits lost during the solution of very complex structures is tolerable. Generally, the loss of 10 digits or more indicates a highly dependent system of equations that requires modifications to be made to the structural model.

The user has the option of requesting several types of ill-conditioning checks to be performed on the structural stiffness matrix during execution of the MERGE processor. These checks provide the user with indicators of the level of ill-conditioning prior to solving the equations by the CHOLESKY processor. Use of these facilities, particularly for complex models, has shown to be effective in the identification of ill-conditioned regions of a model prior to entering the solution phase.

The method used by the MERGE processor in performing the conditioning checks is based on nodal stiffness matrices. That is, the stiffness components of all elements directly associated with a node are used to form a local stiffness matrix. A matrix of this type is created for each node in the finite element model. Based on the minimum and maximum eigenvalues for each matrix, in addition to user-selected ratios, various diagnostic tests are performed. An error message is issued, for example, when the minimum eigenvalue is less than or equal to zero or when

the maximum eigenvalue is greater than the minimum eigenvalue times 10^{14} . Interpretation of the latter situation is that the range in stiffnesses corresponding to the nodal freedoms requires approximately 14 digits of accuracy. Each of the foregoing conditions would obviously introduce significant error into the solution of the equilibrium equations.

5.3 MASS DISTRIBUTION MODEL (User's Manual; secs. 138, 238 and appendix C)

The process of modeling a mass distribution involves assembling an array of finite elements that represent the physical mass distribution. In general, the total mass model is comprised of nodes, structural finite elements, mass finite elements, concentrated masses, and various fuel and payload distributions. All these components can be used, but need not be present in a particular model. Descriptions of all the types of mass distribution input data are presented in section 138 of the User's Manual.

As with the structural model, the mass model is identified by an integer SET number which must be the same as that used to identify the corresponding structural model. If only a weight analysis is to be performed, the mass model SET number may be different from the structural model SET number. There can be 36 independent mass sets in one job. Corresponding to each SET, 100 different mass distributions can be specified; each identified by a CONDITION number.

The various components of a mass distribution model are discussed in the following sections:

<u>Section</u>	<u>Description</u>
5.2	Structural Model (nodes, boundary conditions and stiffness finite elements)
5.3.1	Mass Finite Elements
5.3.2	Concentrated Masses
5.3.3	Fuel and Payload
5.3.4	Mass Modeling Considerations
5.3.5	Mass Options

Formatted printout of the mass-distribution model data (nodes, coordinate systems, elements and boundary conditions) can be generated by including appropriate "PRINT INPUT" statements in the execution-control deck (see sec. 6 and the User's Manual; secs. 206, 246 and 238). Pictorial plots of the node and element grids can also be created using the GRAPHICS postprocessor (see sec. 6.3).

5.3.1 Mass Finite Elements

Mass distribution characteristics that are not represented by the structural finite elements can be idealized by mass finite elements interconnected at the structural nodes or at additional nodes independent of the structural model. One exception to the option of using independent "mass" nodes is when a Guyan reduced mass matrix (Mass Option 4) is generated. In this case, all mass elements must be connected to active structural nodes. In all other cases, the mass distribution model may be independent of the structural model.

A maximum of 32 767 mass finite elements can be used to define a total model or a substructure component (see sec. 5.9) by a MASS data set. These elements represent the mass distribution only and do not affect the stiffness characteristics of the model. Each element-definition data record contains the following sequence of information:

- a) Eltype -- Element type name or identifier (e.g., ROD, BEAM, or PLATE)
- b) Fx ----- Element input format code (e.g., F2)
- c) Userid -- User selected element number preceded by the letter N, or an alphanumeric element identification (e.g., N101 or RIB-15)
- d) Nodes --- List of node numbers that define the element
- e) Plist --- List of element property values (e.g., 0.5 for a half inch thick plate element)

If "Fx" and "Userid" are not input, appropriate default values are used automatically. For example, if "Userid" is not input, the element is assigned a number corresponding to its input sequence number and an identification that represents the element type (e.g., BEAM). All the other data items are required.

The input format code provides the option of defining the element mass properties in terms of density, weight, or the weight and center of gravity (tapered elements only). For the density and weight options, the user may specify the element cross section or thickness properties, whereas with the weight and center of gravity option, the system automatically assigns appropriate cross sectional areas and thicknesses.

Options are provided for generating sequences of elements provided they are the same type and have the same properties. A

grid of plate elements, as shown in figure 5-13, can be created by the following data records. For this case, the density option is used.

```

PLATE F1 N101 1 2 5 4 .1 .5 TO N103 3 4 8 7 /
**2 0 0 200 4 4 4 4 .0 .0 0 200 4 4 4 4 /

```

The mass elements provided by ATLAS and the characteristics of each element type are described in the following subsections. The geometric shape, the number of nodes, and the property input options for each element type are shown in table 5-4.

Detailed descriptions of the ATLAS mass finite elements are presented in appendix C of the User's Manual.

5.3.1.1 SCALAR Element

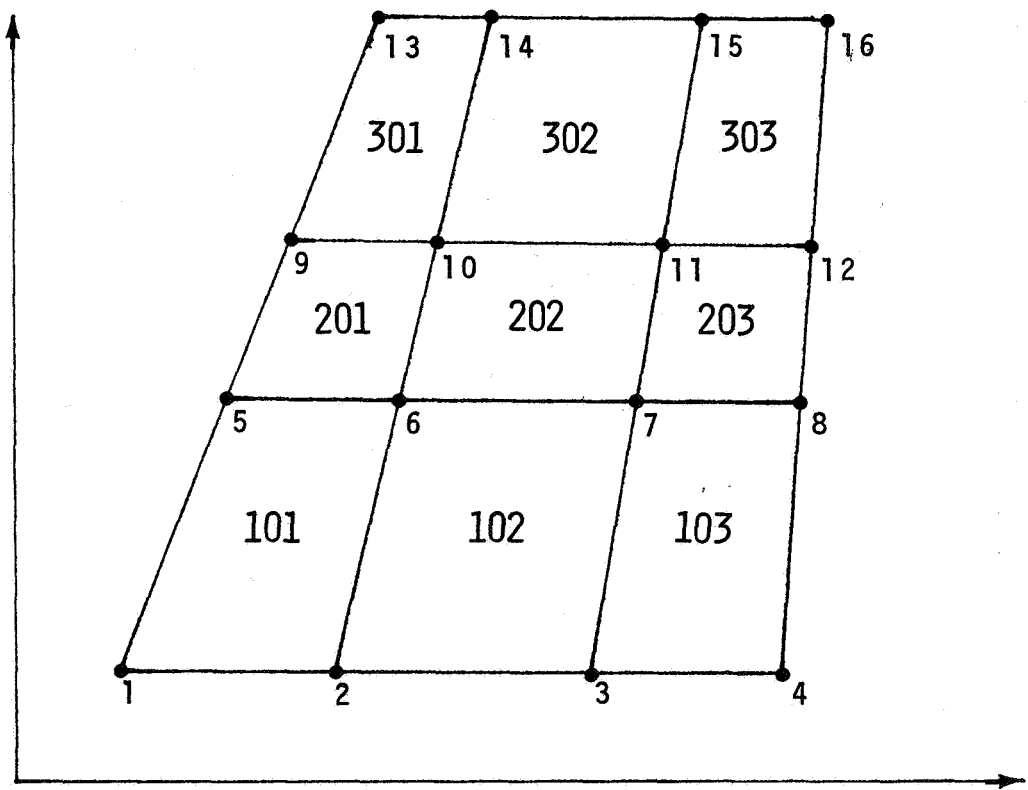
This element allows the representation of mass acting at a single node. The mass at a node is described in terms of its weight, and optionally, it can be assigned three weight moments of inertia and three cross products of inertia. The element mass matrix, in general, has six degrees of freedom.

5.3.1.2 ROD and BEAM Elements

These elements allow linearly varying one-dimensional mass distributions to be defined in terms of density and cross section properties, weight and cross section properties, or weight and center of gravity. These options are used to model mass properties when only the weight, center of gravity, and the extent of the mass are known. The BEAM element differs from the ROD in that cross-sectional area moments and offsets can be defined for the BEAM. These additional features are used primarily in conjunction with the Guyan reduced mass matrix (Mass Option 4) to provide a better description of the element inertia characteristics. In addition, the mass may be transferred to an offset structural node. In general, the mass matrix for the BEAM element contains six degrees of freedom at each node, whereas the mass matrix for the ROD element contains only three translational degrees of freedom.

5.3.1.3 PLATE Element

This element is used to represent a mass distribution when either the weight or the weight and center of gravity of a two-dimensional region are known. A plate is described in terms of its weight, its weight and center of gravity, or its density and thickness. If the weight and center of gravity are specified, the system will taper the plate thickness such that the center of







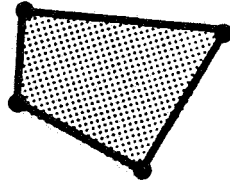

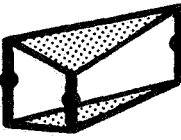
9 PLATE ELEMENTS

DENSITY = .1

THICKNESS = .5

Figure 5-13. Example Plate Element Grid

Table 5-4. The ATLAS Mass Elements

ELEMENT NAME	NODES	GEOMETRIC SHAPE	PROPERTY INPUT OPTIONS						
			DENSITY	THICKNESS	CROSS-SECTIONAL AREAS	AREA MOMENTS OF INERTIA	WEIGHT	WEIGHT MOMENTS OF INERTIA	CENTER OF GRAVITY
SCALAR	1						•	•	
ROD	2		•		•				
					①		•		
							•		•
BEAM	2 (3)		•		•	①			
	4 (5)				①	①	•		
							•		•
PLATE	3 - 9		•	•					
				①			•		
							•		•
SPAR	2		•	•	•				
COVER	3 - 9		•	•					
				①			•		
							•		•

① Default value may be used or property may be input

② Rigid element components (offsets and rigid connections) are illustrated by :



gravity is satisfied. This option is limited to center of gravity points that are near the plate centroid because the plate surfaces are assumed to remain flat when the plate is tapered.

A plate element can have three to nine sides except when a Guyan reduced mass matrix (Mass Option 4) is desired. In this case, the element is limited to a triangular or quadrilateral shape. The plate element mass matrix used in the Guyan reduction contains three translational degrees of freedom at each node.

5.3.1.4 Built-up Elements

The built-up mass elements are identified as the SPAR and COVER elements. These elements are comprised of two or more of the basic elements and are used for modeling specific structural components. For example, in modeling box beam type construction as found in wing structures, the SPAR element would be used to model the spars and ribs, whereas the COVER element would be used to model the surfaces. The geometric description of this element is based on the concept of a "mid-surface" node. These nodes define the mid-surface of the box beam with a "delta Z" coordinate that locates the upper and lower surfaces. The mass matrix for these elements is computed at the mid-surface points.

SPAR Element

This element is comprised of a quadrilateral PLATE element of constant thickness representing the web, and two linearly tapered ROD elements representing the upper and lower caps. At each end of the element a massless rigid post connects the mid-surface points to the upper and lower surfaces.

In contrast to the other mass elements, this element is described by a material density, web thickness, and cap areas. The option to input a weight and center of gravity for the element is not available. The mass matrix is computed at the mid-surface nodes with five degrees of freedom at each node (the local RZ freedom is inactive).

COVER Element

This element is built-up from two PLATE elements separated by massless rigid posts connecting the mid-surface points to the upper and lower surfaces. The mass of the COVER element is described by the individual surfaces in terms of weight, weight and center of gravity, or thickness and density. As with the PLATE element, if the center of gravity is specified, it must be near the plate centroid because the surfaces are assumed to remain flat when the element is tapered. A COVER element may have three to nine sides except when a Guyan reduced mass matrix

(Mass Option 4) is desired. In this case, the element is limited to a triangular or quadrilateral shape. The resulting element mass matrix has five degrees of freedom at each node. As with the SPAR, the local RZ freedom is inactive.

The SPAR and COVER elements are intended to be used together with element connectivity at the mid-surface nodes. However, if convenient, these elements may be used in conjunction with the other types of mass elements.

5.3.2 Concentrated Masses

In addition to the structural model and mass element model, the capability of defining concentrated masses is provided. A concentrated mass represents a component or region of structure that is concentrated at a single point when computing a mass matrix. This point is generally the center of gravity of the component or structural region. In addition to the weight and center of gravity, the user can define up to six weight moments of inertia for each concentrated mass. These inertias are described relative to a coordinate system that is located at the center of gravity of the concentrated mass, and is parallel to either the GLOBAL or a user-defined local reference frame.

In general, the node defining the concentrated mass center of gravity will not coincide with a structural node. Therefore, the mass matrix must be transferred from the center-of-gravity node to the structural node. This is performed automatically by the system whenever an "offset" node is used in defining a concentrated mass. The offset node represents a retained structural node to which the concentrated mass is transferred. The resulting mass matrix will, in general, be a 6 x 6 matrix containing translational mass terms, first moment coupling terms, and a 3 x 3 matrix partition with inertia terms in the analysis frame of the structural node. Whenever a concentrated mass is specified at a retained structural node, no other mass from the structural or mass element model is lumped at that node. This characteristic of concentrated masses distinguishes them from the SCALAR mass elements, which are lumped in conjunction with the other mass and structural elements.

When mass matrix option 2 or 3 (diagonal or non-diagonal mass matrices) is used and only translational degrees of freedom are retained, the weight moments of inertia need not be defined because no inertia terms are associated with translational degrees of freedom.

The concentrated mass options can be used in a number of different ways to model mass distributions. Concentrated masses may be used in conjunction with the structural and mass element

model to represent certain components such as nacelles or landing gears. Multiple subsets of concentrated masses can be defined, thereby providing the capability of generating mass matrices for multiple weight conditions. Defining the variable weight components as separate subsets of concentrated masses eliminates the need of redefining the constant mass components. Concentrated masses can also be used to represent the total mass distribution. This is particularly useful when only the weight and center of gravity of a structural region or component are known and there is insufficient structural model detail to represent the mass distribution. This would be the case when a beam "stick" model of a wing is being used and the weight and center of gravity of the various "wing panels" are known. The weight of each panel would be concentrated at its center of gravity and "offset" from a retained structural node on the beam representing the wing stiffness characteristics.

In addition to representing actual mass distributions, the concentrated mass option may be used to input directly a known diagonal mass matrix. In this case, the concentrated masses are defined with the required properties directly at the retained structural nodes.

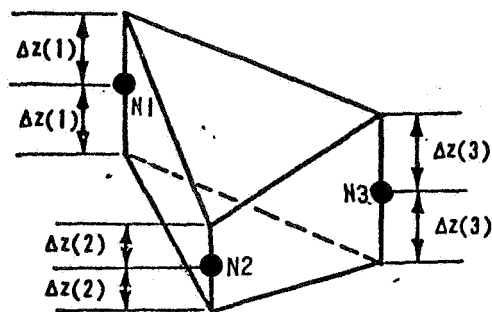
5.3.3 Fuel and Payload

In addition to the structural and fixed weight items, the total mass distribution for a flight vehicle may include a number of fuel and payload distributions representing various gross weight conditions. Fuel and payload can be modeled using mass finite elements and/or concentrated masses. However, if more than one distribution is required, it is generally more convenient to model these mass properties using the fuel and payload data options. These options provide the capability of defining multiple fuel and payload distributions, and then selecting the one desired for each mass matrix. This approach requires less execution time and cost because the structural and fixed weight portion of the mass distribution are not recalculated for each fuel and payload condition.

5.3.3.1 Fuel Data

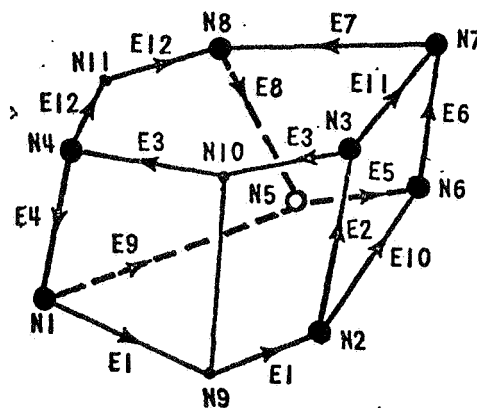
In defining the fuel distribution, the user must provide three basic blocks of data: 1) the fuel tank geometry, 2) the fuel loading and/or usage sequences, and 3) the CONDITION data which define the vehicle attitude and amount of fuel for the condition.

The fuel tank geometry is defined in terms of sections which are represented by the two basic shapes shown in figure 5-14. Each of these shapes is defined in terms of node points which may



TYPE1 Shape

Three or four mid-surface nodes N1,N2,N3 (and N4) are used. Addition of the respective Δz coordinates to the nodal z-coordinates defines the upper surface, whereas subtraction defines the lower surface. The Δz directions are defined by the input nodal z-axes which need not be parallel. If 4 nodes are used, they must be input sequentially in either a clockwise or counterclockwise direction.



TYPE2 Shape

The basic solid is defined by 8 nodes. These nodes define 12 edges (E1-E12) and directions thereof as shown in the figure. An intermediate node may be specified on each edge. The 8 unique corner nodes must be input first followed by any edge nodes. The first 8 nodes define two opposite sides of the solid. N1-N4 may be input clockwise or counterclockwise, however, N5-N8 must be ordered so that each of the node pairs (N1,N5), (N2,N6), (N3,N7) and (N4,N8) lies on a separate edge. The 1 to 12 edge nodes must be input in an order consistent with the edge numbers. If only some of the edges have nodes, zeros must be input in the appropriate locations so that a total of 20 integers are input. The nodal input for the example shown would be N1,N2,...,N8,N9,0,N10,0,0,0,0,0,0,0,N11.

Figure 5-14. Fuel Tank and Cargo Hold Basic Shapes

be nodes used for the structural model or nodes defined strictly for describing the fuel tanks. In defining the tanks, attention should be paid to logical node numbering schemes that allow use of the tank-section generation capabilities, as well as the node generation capabilities. In addition to the basic geometry of each fuel tank, options are provided for specifying a fuel density other than the default value of .0294 lbs/in³, and what percent of the fuel tanks contain usable fuel.

The actual fuel distribution is not defined explicitly. Instead, the sequence of loading and/or using fuel is specified from which the system calculates the desired fuel weight. The resulting distribution is used to generate the required mass matrices. The loading and usage of fuel are defined by the fuel "MANAGEMENT" data subset. These data records specify the amount of fuel in each tank (loading commands), the sequence of tank usage, and the amount of fuel used out of each tank prior to using another tank (usage commands). A number of usage options is provided, including the sequential use of fuel tanks, simultaneous use from 2 or 3 tanks, simultaneous but with different rates of usage, and the transfer of fuel from one tank to another. In all cases, the fuel usage automatically stops when a specified end condition is reached. This may occur when a specified fuel weight remains, when a specific amount has been used, when a tank is empty, or when a defined ratio of fuel in the tanks has been reached.

After the tank geometry and fuel management procedures are established, the weight conditions for generating fuel mass data are defined by the ATTITUDE and CONDITION data subsets. The fuel conditions are specified by defining the vehicle attitude in terms of roll, pitch, and yaw angles measured from the GLOBAL reference frame, a vector normal to the surface of the fuel, the total fuel weight for the conditions, and the fuel management sequence to follow in achieving the desired weight.

5.3.3.2 Payload Data

As with the fuel distribution, the payload distribution is defined by three basic blocks of data; 1) the cargo hold and seat geometry, 2) the payload loading sequence, and 3) the CONDITION data defining the amount of cargo and number of passengers on board.

The passenger seat locations are defined in the same manner as nodes, wherein a seat number and its X, Y, Z coordinates are specified. In defining the seats, options are provided for using local reference frames, for generating rows of seats, and for repeating rows of seats. Use of these options, in general, allows all seat locations to be defined with a minimum of input

data. Definition of the cargo hold geometry is the same as the definition of fuel tank geometry. Each hold is defined in terms of sections which are represented by the two basic shapes shown previously in figure 5-14. Options for section and node generation are the same as discussed for the fuel tanks.

As for the fuel data, the actual payload distribution is not defined explicitly. A sequence of loading commands is defined to specify the desired passenger and cargo loading sequence. These data are used along with the cargo density, passenger weight, and weight moment of inertia data to distribute the payload.

After the geometry and loading sequences are defined, the payload conditions are defined in terms of the number of passengers, cargo weight, and loading sequence for distributing the passengers and cargo.

5.3.4 Mass Modeling Considerations

The primary responsibility of the engineer assembling a mass model is to provide an idealized representation of the actual mass distribution that is detailed enough to give accurate results but not so large as to become prohibitive in terms of setup time or execution cost. The principal factors involved in this responsibility are discussed in the following section and in section 5.2.5, (Fundamental Modeling Considerations).

5.3.4.1 Level of Detail

The amount of detail required for the mass model is generally determined by the overall level of detail of the analysis and the amount and nature of the available mass data. A large substructured model would require parallel mass substructures, each with many structural and mass elements to represent the mass distribution, whereas the mass distribution for a "stick" model of a wing could be represented with 10 to 15 concentrated masses. Another factor in determining the level of detail for the mass model is the nature of the mass distribution itself. If large regions of the structure have reasonably uniform mass distribution, they should be represented by single elements. Nothing is gained by dividing a large area into many elements unless the mass distribution is such that this is necessary. It is usually not necessary to model each item separately even in large detailed models. Multiple components in the same geometric region of the structure can be lumped together and modeled with fewer elements.

In general, a model will have varying degrees of detail for different sections of the structure. For example, a wing structure may be very detailed and use the structural elements

and non-optimum factors, fuel data, and mass elements for representing the systems and secondary structure. On the other hand, the body may be a stick model and need only be represented by a series of concentrated masses.

5.3.4.2 Choice of Elements

Closely related to the level of modeling detail is the choice of elements used to represent the mass distribution. The ATLAS concentrated masses and mass elements provide a variety of options in representing one-, two-, and three- dimensional mass regions. The SCALAR element may be used to represent three-dimensional solid mass regions such as large forgings and major system components that can be lumped at a point in the structure. One-dimensional components, such as long wire or duct runs and spars for which the mass distribution can be considered along a line, should be modeled using ROD, BEAM or SPAR elements. Generally, the ROD element provides an acceptable level of detail and is the simplest of the three elements.

The two-dimensional PLATE element is the most versatile element in that it can be used to model a wide variety of mass distributions. It can be used to represent a large uniform distribution of mass such as an exterior finish, or a wing idealized as a single element. It can also be used to model detailed components of a built-up skin panel, wing leading and trailing edge structure, flaps, or a complete horizontal tail. In general, the PLATE element is used to represent any two dimensional component or region when the weight and geometric bounds of the weight are known.

The choice of elements for a particular model is influenced by many factors including the structural model characteristics, the detail of the overall analysis, the information available about the mass distribution, and the engineer's experience.

No specific rules or formula can be stated for establishing the level of detail or element selection necessary for a sufficiently accurate analysis in a given situation. The engineer must consider the overall analysis and the nature of the mass data available, and must establish the level of detail of the mass distribution accordingly.

5.3.5 Mass Options

In addition to the considerations discussed above, the required type of calculated mass data (e.g., mass matrices, detailed weight statements and fuel/payload data) can have an influence on the mass distribution model. The MASS processor, for example, provides the capability of calculating three types

of reduced mass matrices using OPTIONS 2, 3 and 4; diagonal, non-diagonal, and Guyan reduced, as discussed in the following sections. Mass OPTION=1 is used for calculating weights and inertias of the individual elements and the total mass-distribution model.

All calculated mass data can be printed by including appropriate PRINT OUTPUT (MASS) statements in the ATLAS execution-control deck (see sec. 6 and the User's Manual; sec. 238). In verifying a mass model, for example, the weight and c.g. can be printed for the total model or for user-selected components and regions of the model. Detailed weight-statement options allow the user to itemize and label the selected weight data in a tabular printout.

5.3.5.1 Diagonal Mass Matrix (OPTION=2)

The diagonal mass matrix contains only the lumped translational mass terms and the diagonal inertia terms. The first moment and cross product of inertia terms are ignored. The matrix is generated by lumping each incremental mass " ΔM " of the model directly at the closest retained node. If rotational degrees of freedom are retained, the mass moments of inertia for the mass lumped at each node are computed. The resulting masses and inertia are assembled to form a diagonal mass matrix. This type of mass matrix is the least expensive to generate and can be used when the grid of retained nodes is uniformly distributed such that the center of gravity of the mass to be lumped at a node is close to that node. This minimizes the effect of ignoring the off-diagonal first moment terms. If concentrated masses are used, all inertia terms are ignored except for the diagonal terms. In this case, the retained structural nodes should coincide with the mass centers of gravity. If only translational degrees of freedom are retained and the accuracy of a Guyan reduced mass matrix is not needed, the diagonal mass matrix option should be used.

In general, use of the diagonal mass matrix requires a reasonably fine nodal grid where the retained nodes are located uniformly throughout the structure. If these criteria are satisfied, the diagonal mass matrix provides an inexpensive and accurate option for generating the required mass distribution data.

5.3.5.2 Non-Diagonal Mass Matrix (OPTION=3)

When generating a mass matrix for a "stick" model or a coarse-grid model, the non-diagonal mass matrix generally provides more accurate results because the rotational inertia and first moment coupling terms are included in the analysis. This

matrix is generated like the diagonal matrix in that each incremental mass "dM" of the model is lumped at the closest retained node. The primary difference is that instead of lumping the mass directly at the retained node, the data mass to be lumped at a given node is accumulated, and its center of gravity and six mass moments of inertia are computed. The resulting mass terms at the center of gravity are then transferred to the retained node and assembled directly to form the reduced mass matrix.

When using a non-diagonal mass matrix, attention should be given to preserving the rotational inertia characteristics of the structure when the mass model is defined. If concentrated masses are used, the weight moments of inertia corresponding to the retained rotational degrees of freedom should be defined.

Use of the non-diagonal mass matrix option allows the mass and inertia characteristics of the structure to be represented accurately even when the grid of retained nodes is somewhat coarse or it does not enclose the entire structure. In all cases, the rotational degrees of freedom that couple with the retained translational degrees of freedom must be retained in order to preserve the inertia characteristics of the model.

5.3.5.3 Guyan Reduced Mass Matrix (OPTION=4)

A detailed vibration analysis of a structural component may require the increased accuracy and cost of generating a Guyan reduced mass matrix. If this option is selected, there are certain restrictions placed on the mass distribution model. All mass elements must be connected to structural nodes, the fuel and payload options are not available, and the mass PLATE and COVER elements are limited to triangles and quadrilaterals. Any concentrated masses are merged along with the element mass matrices.

A Guyan reduced mass matrix is generated by calculating an elemental mass matrix for each element in the structure, merging the elemental matrices with any concentrated masses to form the gross mass matrix, and then using the Guyan reduction procedure (ref. 5-3) to reduce the gross mass matrix to form the final reduced mass matrix. The elemental mass matrices are calculated as lumped non-diagonal matrices formed by proportioning the element mass to each active element node, computing the center of gravity and inertias of the proportionate mass, and then transferring the mass and inertia to the corresponding node. This procedure is illustrated in figure 5-15 for an offset beam. The BRICK element is an exception to the procedure in that its mass matrix is a consistent mass matrix (ref. 5-4) based on the

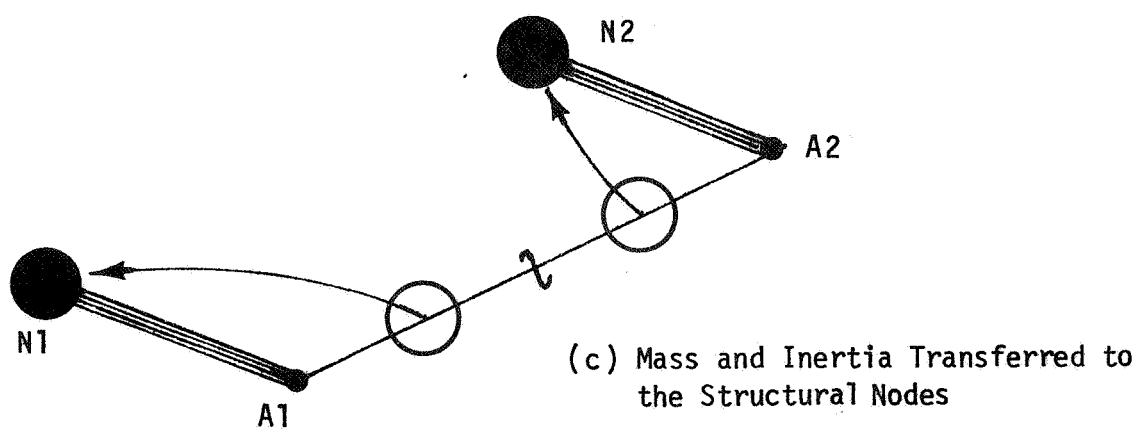
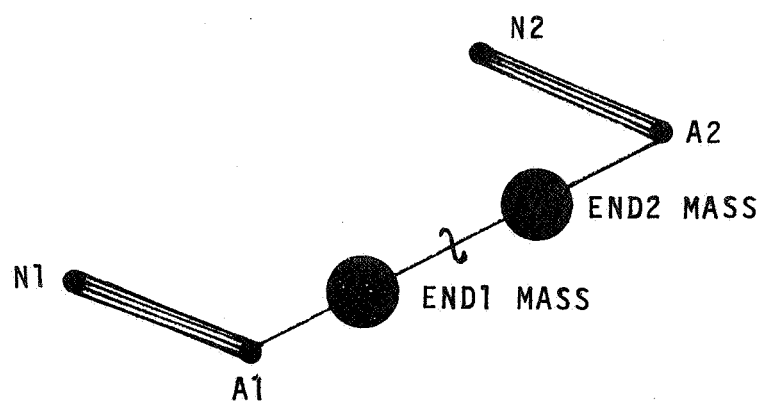
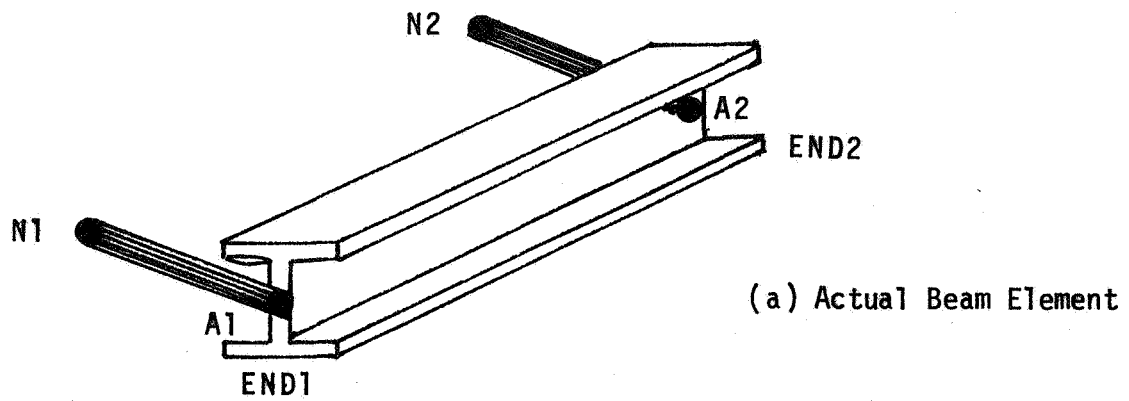


Figure 5-15. Offset Beam Mass Lumping

integration of the assumed displacement function used for the element stiffness matrix formulation. Merging of the elemental mass matrices and the Guyan reduction are performed by including a PERFORM M-REDUCE statement in the execution-control deck (see sec. 6 and the User's Manual; appendix E).

The Guyan reduced mass matrix formulation is expensive and should be used only when the diagonal or non-diagonal options do not yield sufficiently accurate results, or when BRICK elements are used in the model. The only mass matrix option available for BRICK elements is the Guyan reduced mass matrix.

5.4 STATIC LOADS (User's Manual; secs. 134 and 234)

This section describes the various types of static loads that can be applied to a structural model. These loads can be used to perform static stress analyses, bifurcation buckling analyses (general instability), and automated structural resizing by the ATLAS System.

All loading is associated with a particular structural model (a SET) and one of its groups of boundary conditions (a STAGE). In a substructured analysis, loads can be applied only to the lowest level substructures (see. sec. 5.9).

Discussions of the various types of static loads are presented in the following sections.

<u>Section</u>	<u>Loading Type</u>
5.4.1	Support Displacements
5.4.2	Nodal and Element-Distributed Loads
5.4.3	Rotational Inertia Loads
5.4.4	Thermal Loads
5.4.5	ATLAS/FLEXSTAB Interfaced Loads

All loads input with the same loadcase label, regardless of type, are automatically cumulated. A loadcase can optionally be defined as a linear combination of other loadcases. Detailed descriptions of the LOADS data are presented in section 134 of the User's Manual, whereas descriptions of how the various types of loads are processed are presented in the ATLAS "LOADS Module Theory" document.

Formatted printouts of selected loadcase input data, as well as the cumulated nodal loads and applied load resultants are generated by including appropriate "PRINT" statements in the ATLAS execution-control deck (see sec. 6 and the User's Manual; sec. 234).

5.4.1 Support Displacements

By means of "SUPPORT DISPLACEMENT" data, a user can prescribe motion for supported degrees of freedom. These displacements are loadcase dependent and can be combined with any other type of static loading in a given loadcase. It should be noted that support displacements are specified relative to the nodal analysis frames.

Support displacements may be input in either the BC data set or the LOADS data set (or both). Input formats for these data are identical in both data sets.

If a nonzero displacement value is not specified for a supported freedom, it is considered as being rigidly fixed.

5.4.2 Nodal and Element-Distributed Loads

"NODAL LOAD" data provide the means for direct specification of a concentrated force or moment acting at a node. Only nonzero force or moment components need be specified. Any local reference frame defined in the NODAL data set can be used to specify the direction of action of the nodal loads. Multiple loads applied to a certain freedom for a given loadcase are cumulated.

"ELEMENT LOADS" data are used to specify distributed pressure loading acting on stiffness finite elements. The loading may act in any direction and is specified per unit length for linear elements (ROD, BEAM, SROD) or per unit surface area for plate elements (PLATE, COVER, GPLATE, SPLATE, CPLATE, CCOVER). The direction of action may be specified relative to either the GLOBAL frame or the element reference frame for all elements except the BRICK, for which the element reference frame must be used. Except for the BRICK, the nodal forces resulting from the specified element loads are a simple set of statically equivalent forces. Consistent nodal forces are calculated for the BRICK. Multiple loads applied to an element for a certain loadcase are cumulated.

5.4.3 Rotational Inertia Loads

Inertia loads resulting from the rotation of a structure about a stationary axis can be calculated by the ATLAS System. The state of rotation is defined by the "INERTIA LOADS" data subset whereby the axis of rotation and the positive sense of rotation are specified by two nodes. Multiple axes of rotation may be used in a single analysis. For each axis of rotation, any number of loadcases may be defined by specifying different angular velocity and/or angular acceleration components. The

following types of mass components contribute to the inertia loads:

- a) Stiffness finite elements
- b) Mass finite elements
- c) Concentrated masses

Inertia loads are automatically combined with any other types of loads associated with the same loadcase identifier.

5.4.4 Thermal Loads

"THERMAL LOADS" data allow the user to specify changes in temperature which result in stresses and deflections. These thermal loads (temperature changes) may be specified either as nodal loading or as element loading. The loads must be applied as element loading when a thermal load discontinuity occurs at a node or when there are gradients through the depth of an element at a node (e.g., a gradient through the thickness of a GPLATE). If both nodal and element thermal loads are specified for the same point in the model, only the element thermal loads are used.

The ΔT thermal loads are changes with respect to the element base temperature, T_0 , which is the value of "Tcode" in the element definition record of the STIFFNESS data set. The applied thermal strains, ϵ , are calculated by ATLAS in the following manner:

$$\epsilon = \epsilon_{Tf} - \epsilon_{Ti}$$

where

$$\epsilon_{Tf} = \epsilon_T \text{ from material table at final temperature } (T_0 + \Delta T)$$

$$\epsilon_{Ti} = \epsilon_T \text{ from material table at initial temperature } (T_0)$$

The user must be careful to define T_0 , ΔT and the ϵ_T values in the MATERIAL data set such that the desired thermal strains result. Note that a nonzero ϵ_T at T_0 does not produce any strain loading by itself; a nonzero ΔT must be specified.

To calculate the internal forces resulting from these initial strains, the element stiffness at temperature T_0 is used. An exception to this may be made, as a user option, for the BRICK. This option is selected by the MATERIAL parameter in the EXECUTE LOADS control statement (sec. 234 in the User's Manual). If the VARIABLE material option is selected, the stiffness of the BRICK is calculated using the material properties at the final

temperature ($T_0 + \Delta T$) which may differ from node to node within the element.

Thermal gradients through the depth of a BEAM are specified by input of a THERMAL LOADS data subset and a "DETAIL" data set (sec. 114 in the User's Manual). The gradient through the BEAM in the element y-direction is defined as

$$\frac{\Delta T(+y) - \Delta T(-y)}{dy}$$

where the ΔT s are specified in the LOADS data and "dy" is defined by the DETAIL data set. The gradient through the beam in the element z-direction is defined similarly as

$$\frac{\Delta T(+z) - \Delta T(-z)}{dz}$$

Note that the gradients in the y and z directions are defined independently of each other.

In order not to restrain an element offset from its structural nodes, axial strains are provided in the rigid offsets. These rigid offset strains are calculated using the strain values specified for the element's material and the average of the thermal loads at the offset ends. If a gradient through the element depth exists, the average element temperature difference at the auxiliary node is used at that end of the offset.

5.4.5 ATLAS/FLEXSTAB Interfaced Loads

The ATLAS/FLEXSTAB interface provides the capability of applying those nodal loads (airloads) generated by the FLEXSTAB System (ref. 5-5) to an ATLAS structural model. In using this capability, the following restrictions apply to the structural model and the analysis procedure.

- a) The model may not be substructured,
- b) All loaded freedoms must be identified as retained freedoms in the boundary condition (BC) data,
- c) The order of the ATLAS retained freedoms must be the same as that used in the FLEXSTAB analysis,
- d) All retained nodes must have a GLOBAL analysis frame oriented such that the positive X-axis is in the

direction of the free stream flow and the positive Z-axis is upward,

- e) The GLOBAL X-Y plane is the plane of symmetry.

Calculation of displacements and stresses is performed by using a special ATLAS Control Program which reads the FLEXSTAB SDSS loads file prior to using the ATLAS R-STRESS execution-control procedure for performing a stress analysis.

The FLEXSTAB loads file may contain loads for both a symmetric and an antisymmetric analysis. The symmetric boundary conditions are assigned STAGE 1 and the loads are assigned integer loadcase labels 1 through N corresponding to the first through N-th loadcases on the FLEXSTAB loads file. The antisymmetric boundary conditions are assigned STAGE 2 and the loads are assigned loadcase labels 1 through N in the same manner as the symmetric loads. These loadcases are automatically combined with any other ATLAS loadcases that have the same STAGE and loadcase labels.

Further descriptions of the ATLAS/FLEXSTAB System interfaces for performing acroelastic loads analyses are presented in appendix G of the ATLAS User's Manual.

5.5 STRESS (User's Manual; secs. 154 and 254)

A stress analysis generally includes the calculation of nodal displacements, element stresses, internal element nodal forces, and reactions. For cost-effective solution of certain problems, superposition of the calculated displacements and stresses for multiple loadcases is also required. All of these types of calculations are supported directly by the STRESS processor.

5.5.1 Problem Formulation

The model-definition data sets required for performing a basic stress analysis are as follows (see table 5-1):

- a) NODAL -- see sec. 5.2.2
- b) BC (Boundary Condition) -- see sec. 5.2.3
- c) STIFFNESS -- see secs. 5.2.4 and 5.2.5
- d) LOADS -- see sec. 5.4

A STRESS data set is required only if displacements and stresses are to be superimposed; a superposition loadcase is defined as a linear combination of one or more other loadcases. Factors assigned to the component loadcases may be known or unknown. The unknown factors are calculated by using specified constraints on

selected nodal displacements and/or element stresses (User's Manual; sec. 154).

The structural model must be supported (constrained) from any rigid body motion by specifying a sufficient number of supported freedoms in the BC data set (User's Manual; sec. 106).

Displacements, internal element nodal forces, and reactions are calculated relative to the node analysis reference frames specified by the NODAL data set (User's Manual; sec. 146), whereas stresses are calculated relative to the individual element reference frames (User's Manual; appendix B). The calculated stresses and related sign conventions, as well as the orientation of element reference frames for the various types of ATLAS elements are described in appendix B of the User's Manual.

Displacements, stresses and reactions are calculated simply by including a PERFORM STRESS statement in the execution-control deck (see sec. 6). Internal element nodal forces are also calculated when the execution parameter OPT=(1,2,3) is specified in this statement. Plots and printout of these calculated data are generated as discussed in section 5.5.3 (Stress Postprocessing).

5.5.2 Solution Accuracy Checks

The structural-data quality checks provided by ATLAS for detecting and locating errors associated with the structural model were discussed previously in section 5.2.6. In addition to using those data checks, solution accuracy checks should be used in verifying the quality of the problem solution.

Overall checks on the accuracy of a stress analysis are whether the calculated displacements and stresses are reasonable, and how much variation there is in the stresses for adjoining elements. Generally, if the node-to-node displacements or the element-to-element stresses vary significantly, a more refined element grid layout must be used.

The quality of a stress analysis can also be verified by requesting the load-reaction equilibrium check to be performed by the REACTION postprocessor (User's Manual; sec. 248). For each loadcase, the applied loads are added to the computed reaction forces. The resultant six load components are displayed for each loadcase. In general, any significant discrepancy between the applied load and reaction components reflects a poorly conditioned solution. Equilibrium of the loaded structure has not been satisfied adequately. The user is cautioned that the equilibrium check does not include the reactive forces of ground on any SCALAR elements in the structural model. In this case,

the equilibrium check results should balance the resultants of all the SCALAR element internal forces, as printed in the STRESS output.

Overall equilibrium of the loaded structural model must be ensured by supporting (fixing) a sufficient number of kinematic freedoms so that any rigid body motion is prohibited. Additionally, sufficient structural elements or kinematic constraints must be provided so that no mechanisms exist within the structural model. If the equilibrium equations are ill-conditioned and processing is allowed to continue through completion, structural instabilities can be detected by the load-reaction equilibrium check, unreasonable displacements at certain freedoms, or by a high loss of significant digits during solution. Generally, the kinematic freedoms corresponding to the high digit losses, as identified by the CHOLSKY processor, allow the user to identify necessary corrections to be made to the boundary conditions and/or the finite element model.

5.5.3 Stress Postprocessing

Selected printout of the calculated displacements, stresses and internal element nodal forces is requested by including the following appropriate statements in the ATLAS execution-control deck.

```
PRINT OUTPUT (DISPLACE)
PRINT OUTPUT (STRESSES)
PRINT OUTPUT (FORCES)
```

Optional execution parameters in these statements allow the printed-page formats for these data to be specified by the user. Furthermore, the printed data can be limited to only those nodes and elements included in nodal and element subsets as defined by a SUBSET DEFINITION data set (User's Manual; sec. 156). Those stresses, for example, corresponding to a list of user-selected elements, a certain element type, or those elements in a geometric region defining a certain structural component can be printed in a block. Multiple blocks of data corresponding to multiple node/element subsets can be requested. Use of these features, as described in section 254 in the User's Manual, allows for convenient interpretation of the large quantities of stress data generally required.

Printout of the internal forces acting on the separate elements at a particular node can be requested by using a PRINT OUTPUT (FREEBODY) statement in the execution-control deck (User's Manual; sec. 224). The isolated elements, a free-body region of the model, are included in an element subset by input of SUBSET DEFINITION data.

Printout of the calculated reactions, as well as the load-reaction equilibrium check described in section 5.5.2, is requested by using the execution-control statement PRINT OUTPUT (REACTIONS,EQCHK). This statement and its optional execution parameters are described in section 248 in the User's Manual.

The following types of plots can be created by using the GRAPHICS postprocessor to display the data resulting from a stress analysis:

- a) Nodal displacements
- b) Deformed structural grids
- c) Element stress contours
- d) Graphs of element stress versus loadcase

Further discussion of the ATLAS graphical output is presented in section 6.3.

5.6 BUCKLING (User's Manual; sec. 208)

Critical structural buckling loads and modes are obtained by solving an equation of the form $(K+E*G)*\phi=0$, where K denotes the reduced elastic stiffness matrix, G is the reduced geometric (stress stiffening) matrix, E is the eigenvalue (critical loadcase factor) matrix, and ϕ is the eigenvector (mode shape) matrix. The linear bifurcation buckling mode shapes and the critical loadcase factors are calculated by the BUCKLING processor.

5.6.1 Problem Formulation

The model-definition data sets required for performing a buckling (elastic stability) analysis are as follows:

- a) NODAL -- see sec. 5.2.2
- b) BC (Boundary Condition) -- see sec. 5.2.3
- c) STIFFNESS -- see secs. 5.2.4 and 5.2.5
- d) LOADS -- see sec. 5.4

Geometric stiffness calculations are provided only for the ROD, BEAM, PLATE, GPLATE and BRICK finite elements described in section 5.2.4. Therefore, only these element types can be used in defining a structural model for which a buckling analysis is to be performed.

The structural model must be supported (constrained) from any rigid body motion. Those unconstrained nodal degrees of freedom to be represented explicitly in the buckling equations are identified as retained freedoms in the BC data set (User's Manual; sec. 106). Generally, only a portion of the total number

of unconstrained freedoms need be retained to perform an accurate buckling analysis, provided that they are carefully selected. Although the maximum number of retained freedoms is limited to approximately 400 by the BUCKLING processor, it provides an elastic-stability analysis capability for small to intermediate sized problems.

The reduced elastic stiffness matrix used in performing a buckling analysis is generated from the gross elastic stiffness characteristics at the unconstrained (FREE and RETAIN) nodal degrees of freedom, by "reducing out" the FREE freedoms. Similarly, the reduced geometric stiffness matrix is generally created by using a reduction (static condensation) process to "reduce out" the FREE freedoms from the gross geometric stiffnesses at the unconstrained freedoms. The latter reduction process is performed most conveniently by executing the CHOLESKY and MULTIPLY processors as described below.

Ordering of the retained freedoms in a particular sequence is a BC data option that allows for convenient interpretation of the calculated mode shapes in analyzing specific problems; the i -th component of each mode-shape defines the displacement of the i -th retained freedom. Mode shape displacement components are calculated relative to the analysis reference frames specified for the retained nodes by the NODAL data set (see sec. 5.2.1)

For a buckling analysis, the following problem-solution steps must be performed by including appropriate ATLAS statements in the execution-control deck:

- a) Calculate element stresses and the reduced elastic stiffness matrix K ; use of the PERFORM R-STRESS statement provides this (see sec. 6).
- b) Calculate the elemental geometric stiffnesses by including an EXECUTE STIFFNESS statement in the control deck. In this statement, the name of the loadcase is specified, together with a buckling SET number (BSET) that is used to identify the results. The geometric stiffnesses are a function of the calculated stresses for the specified loadcase (User's Manual; sec. 252).
- c) Assemble the gross geometric stiffness matrix partitions associated with the retained freedoms by using the MERGE processor (User's Manual; sec. 242). For example, EXECUTE MERGE(GSTIFF,GK11=11,GK12=12,GK22=22)
- d) Generate the reduced geometric stiffness matrix, G , using ATLAS execution-control statements of the following type:

```
EXECUTE CHOLESKY (SOLVE,K11,DBS,K12)
EXECUTE MULTIPLY (TEMP=[ GK11*DBS ])
EXECUTE MULTIPLY (G=[ GK22-DBS (T) *GK12-GK12 (T) *DBS+DBS (T) *TEMP ])
```

- e) Request a buckling solution by an EXECUTE BUCKLING statement. In this statement, the names of the elastic and geometric stiffness matrices (K and G) are specified, together with the BSET number. Using the NEIGS and NMODES parameters identifies how many eigenvalues and eigenvectors are to be calculated. Generally, only one eigenvalue (the smallest) is of interest when investigating elastic stability. In this case, the Sturm-sequence method of extracting eigenvalues should be used by including the STURM keyword in the execution parameter list (User's Manual; sec. 208).

The buckling mode shapes and the corresponding eigenvalues can be printed by including a PRINT OUTPUT(BUCKLING) statement in the ATLAS execution-control deck (see sec. 6 and the User's Manual; sec. 208). The printed eigenvalues are those factors by which the applied static loading is multiplied to produce buckling (i.e., the loadcase factors that result in critical loads).

Modal displacements can also be plotted using the GRAPHICS postprocessor (see sec. 6.3). The modes are plotted as displacements of the retained nodes included in a nodal subset. An ordered nodal subset may be used to display an associated grid; retained nodes in the ordered nodal subset are displayed in their deformed positions, whereas other nodes are displayed in their undeformed positions. Nodal subsets are defined by input of a SUBSET DEFINITION data set (User's Manual; sec. 156).

5.6.2 Solution Accuracy Checks

Three numerical quality checks of the eigensolution are performed automatically by the BUCKLING processor. The results of these checks are printed automatically in the ATLAS output file.

- a) Eigenvector Orthogonality Check -- A matrix is printed to indicate the state of orthogonality between each pair of requested eigenvectors (modes). Poor orthogonality (nonzero terms) usually indicates that the selected numerical method encountered difficulties in performing an accurate solution.
- b) Equilibrium Check -- Each calculated eigenvalue and associated eigenvector are substituted into the original

buckling equations. Deviations of the resulting residual vectors from a null vector are printed for use as a measure of the validity of the eigensolution.

- c) Ill-Conditioning Check -- The eigenproblem is solved by reducing it to the standard form $D*\phi=E*\phi$. Using the D matrix, the ill-conditioning check is performed in the following steps:
1. The "Norm" of the D matrix is calculated. Absolute values of the elements in the i-th row of this matrix are summed to form $s(i)$; $i=1, n$, and the maximum $s(i)$ is defined as the "Norm".
 2. The absolute value of the i-th calculated eigenvalue $q(i)$ is compared to the "cutoff" value defined as $(\text{Norm}*10^{-12})$. If $q(i)$ is smaller, it is automatically set to zero and a diagnostic message is printed.

A zero eigenvalue is an adequate indicator that an error exists in the mathematical model which should be corrected for reanalysis.

5.7 VIBRATION (User's Manual; sec. 258)

Normal modes of structural vibration are obtained by solving an equation of the form $(K-E*M)*\phi=0$, where K denotes the reduced structural stiffness matrix, M is the reduced mass matrix (inertia forces), E is the eigenvalue (frequency-dependent) matrix, and ϕ is the eigenvector (mode shape) matrix. Optionally, the equation can be formulated in terms of the structural flexibility matrix instead of the stiffness matrix. The undamped mode shapes, frequencies, generalized mass and generalized stiffness are calculated by the VIBRATION processor.

5.7.1 Problem Formulation

The model-definition data sets required for performing a vibration analysis are as follows (see table 5-1):

- a) NODAL -- see sec. 5.2.2
- b) BC (Boundary Condition) -- see sec. 5.2.3
- c) STIFFNESS -- see secs. 5.2.4 and 5.2.5
- d) MASS -- see sec. 5.3

The structural model can be supported (constrained) from any rigid body motion or it can be free to assume one to six rigid body modes of vibration. Those unconstrained nodal degrees of freedom to be represented explicitly in the equations of motion

are identified as retained freedoms in the BC data set (User's Manual; sec. 106).

The reduced stiffness and mass matrices used in performing a vibration analysis are generated from the gross structural stiffness and mass characteristics at the unconstrained (FREE and RETAIN) nodal degrees of freedom, by "reducing out" the FREE freedoms. The mass matrix can be diagonal (mass OPTION=2), non-diagonal (mass OPTION=3), or a Guyan reduced matrix (mass OPTION=4) as described in section 5.3.5. Accurate vibration analyses are generally performed by retaining only a portion of the total number of unconstrained freedoms.

Ordering of the retained freedoms in a particular sequence is a BC data option that allows for convenient use and interpretation of the calculated mode shapes in analyzing specific dynamic problems; the i-th component of each mode-shape defines the displacement of the i-th retained freedom. Mode shape displacement components are calculated relative to the analysis reference frames specified for the retained nodes by the NODAL data set (see sec. 5.2.1).

Normal modes of vibration are calculated by including an EXECUTE VIBRATION statement in the execution-control deck (see sec. 6 and the User's Manual; sec. 258). In this statement, the names of the mass matrix and the stiffness (or flexibility) matrix are specified, together with a vibration SET number (VSET) that is used to identify the results (see the example problem in sec. 8.4). This statement causes an eigensolution to be performed to obtain the natural frequencies and normal modes of vibration, together with matrices of generalized mass and stiffness corresponding to the modes. The generalized matrices can be limited in size by discarding higher frequency eigenvalues and eigenvectors using the NFREQS and NMODES parameters. If the vibration modes are to be interpolated for purposes of obtaining displacements at points other than structural nodes (e.g., aerodynamic control points and planform boundaries), one or more modal subsets must be requested using the SUBSETS parameter.

If a singular stiffness matrix is used, one or more zero eigenvalues are calculated. The associated eigenvectors obtained by VIBRATION will be some combination of rigid body translation and rotation. Although these eigenvectors are perfectly acceptable and correct, it may be desirable to replace them with known rigid motions of the structure. Several options are available for performing this substitution (the URBM, PRBM, and TRBM execution parameters). Through proper substitution, the resulting modes will be orthogonal to the original elastic modes.

The matrices generated by the VIBRATION processor can be printed by including a PRINT OUTPUT(VIBRATION) statement in the ATLAS execution-control deck. Modal displacements can also be plotted using the GRAPHICS postprocessor (see sec. 6.3). The nodes are plotted as displacements of the retained nodes included in a nodal subset. An ordered nodal subset may be used to display an associated grid; retained nodes in the ordered nodal subset are displayed in their deformed positions, whereas other nodes are displayed in their undeformed positions. Nodal subsets are defined by input of a SUBSET DEFINITION data set (User's Manual; sec. 156).

5.7.2 Solution Accuracy Checks

Three numerical quality checks of the eigensolution are performed automatically by the BUCKLING processor. The results of these checks are printed automatically in the ATLAS output file.

- a) Eigenvector Orthogonality Check -- A matrix is printed to indicate the state of orthogonality between each pair of requested eigenvectors (modes). Poor orthogonality (nonzero terms) usually indicates that the selected numerical method encountered difficulties in performing an accurate solution.
- b) Equilibrium Check -- Each calculated eigenvalue and associated eigenvector are substituted into the original equations of motion. Deviations of the resulting residual vectors from a null vector are printed for use as a measure of the validity of the eigensolution.
- c) Ill-Conditioning Check -- The eigenproblem is solved by reducing it to the standard form $D*\phi=E*\phi$. Using the D matrix, the ill-conditioning check is performed in the following steps:
 - 1. The "Norm" of the D matrix is calculated. Absolute values of the elements in the i-th row of this matrix are summed to form $s(i)$; $i=1, n$, and the maximum $s(i)$ is defined as the "Norm".
 - 2. The absolute value of the i-th calculated eigenvalue $q(i)$ is compared to the "cutoff" value defined as $(\text{Norm}*10^{-12})$. If $q(i)$ is smaller, it is automatically set to zero and a diagnostic message is printed.

This numerical check is based on the rationale that if round-off errors have introduced substantial error in

the small eigenvalues, the eigenproblem is ill-conditioned and the mathematical model should be corrected for reanalysis. Generally, the smaller eigenvalues are of primary interest. It should be noted that zero eigenvalues (frequencies) are produced for any rigid-body modes inherent to a free or partially constrained structural model. Any additional zero eigenvalues, however, clearly indicate that an error exists in the mathematical model.

5.8 FLUTTER

Flutter speeds and damping characteristics can be obtained for a structure moving through a fluid by solving an equation of motion of the form

$$[\omega^2 M - K(1 + ig) + 1/2 \rho V^2 Q(k, \bar{M})] \{q\} = \{0\}$$

where Q represents the oscillatory aerodynamic or hydrodynamic forces experienced by the structure at a reduced frequency k and Mach number \bar{M} . The reduced frequency of oscillation is related to the freestream velocity, V , by the expression $k = \omega b / V$, where ω is the circular frequency of oscillation and b is a reference length.

The flutter equation is formed by calculating a mass matrix that represents the inertia properties of the structure, a stiffness matrix that represents the stiffness properties of the structure, and a set of airforce matrices that represent the aerodynamic forces acting on the structure as it oscillates at constant amplitude while moving through a fluid at constant velocity.

5.8.1 Problem Formulation

The flutter equation can be formed according to any of several theoretical approaches by using different sequences of ATLAS execution-control statements. Consequently, the following discussion refers both to input data for the ATLAS preprocessors, and to data that are input by the execution-control statements described in section 6. Reference should also be made to the example problem in section 8.5.

5.8.1.1 Forming the Flutter Equation - Basic Approach

The most commonly used method of forming the flutter equation requires the following steps to be performed.

- a) Form the mass and stiffness (or flexibility) matrices representing the structural properties in physical

coordinates. These operations are performed most conveniently by using the standard ATLAS execution-control procedures described in section 6.

- b) Calculate the normal modes of vibration in still air, eliminating higher-frequency modes that are unlikely to contribute significantly to the flutter speeds of interest. The selected mode shapes, $[\phi]$, are used as generalized coordinates for which a generalized mass matrix and stiffness matrix are calculated by the VIBRATION processor (see sec. 5.7).
- c) Interpolate displacements constituting the generalized coordinates from the retained structural nodes to the aerodynamic control points pertaining to the selected aerodynamic theory by using the INTERPOLATION processor (see sec. 5.8.2).
- d) Calculate matrices of generalized airforces for a number of discrete reduced frequencies. Several theoretical approaches are provided by the ATLAS aerodynamics processors as described further in section 5.8.3. Namely, these capabilities are as follows:
 - 1) AF1 -- Modified strip theory
 - 2) DUBLAT -- Doublet lattice theory
 - 3) RHO3 -- Assumed pressure modes
 - 4) MACHBOX-- Supersonic Mach-Box theory
- e) Interpolate the generalized airforces with respect to reduced frequency to obtain a larger number of generalized airforce matrices using the ADDINT processor (see sec. 5.8.5).
- f) Modify the generalized mass and stiffness matrices in the flutter equation by introducing structural damping, by further eliminating generalized coordinates, and by defining the fluid density using the FLUTTER processor (see sec. 5.8.6).

The flutter equation is then solved at each reduced frequency, and the speeds at which flutter occurs are determined.

5.8.1.2 Forming the Flutter Equation - Alternative Approaches

Alternative methods of forming the flutter equation, which are variations of the foregoing basic method, are outlined below.

Variation 1:

The flexibility effects of the higher-frequency vibration modes that are not included in the set of generalized coordinates can be included in the generalized airforces by using the following sequence of steps.

- a) Calculate the normal modes of vibration, and the generalized mass and generalized stiffness, having eliminated the higher-frequency modes that are unlikely to contribute significantly to the flutter speeds of interest by using the VIBRATION processor (see sec. 5.7).
- b) Generate aerodynamic influence coefficients, AICs, representing the aerodynamic force on a retained structural freedom due to a unit displacement of another retained structural freedom. This function is performed by using the INTERPOLATION processor plus one of the following processors: AF1, DUBLAT, MACHBOX, or RHO3.
- c) Generate generalized airforces that include the flexibility effects of the truncated modes using the FLEXAIR processor (see sec. 5.8.4).
- d) Interpolate the generalized airforces as before, using the ADDINT processor (see sec. 5.8.5).
- e) The flutter equation should not be modified prior to its solution.

Variation 2:

If suitable AIC matrices are available from a prior analysis, they may be reused by executing steps (a), (c), (d), and (e) of variation 1.

Variation 3:

If the normal modes of vibration $[\phi_R]$ and generalized airforce matrices $[Q_R]$ for a reference analysis are available, $[\phi_R]$ can be used as reference modes in the solution of a closely related problem by doing the following.

- a) Form the mass and stiffness (or flexibility) matrices, $[m_p]$ and $[k_p]$, representing the structural properties in physical coordinates. These operations are performed most conveniently by using the standard ATLAS execution-control procedures described in section 6.

- b) Form the generalized mass and stiffness matrices using

$$[M_p] = [\phi_R^T][m_p][\phi_R]$$

$$[K_p] = [\phi_R^T][k_p][\phi_R]$$

and use $[Q_R]$ as the generalized airforces.

Variation 4:

The flutter equation can be formed directly using the structural mass and stiffness matrices, and the AIC matrices, provided that the matrices are not partitioned and they are written in formats that are acceptable to the FLUTTER processor. This variation generally requires some SNARK statements in the ATLAS execution-control deck.

5.8.1.3 Restrictions on the Structural Model

Regardless of the method used to create the flutter equations of motion, certain conditions must be satisfied.

- a) The GLOBAL X-axis must be parallel to the freestream direction.
- b) The analysis frames of the retained nodes must be oriented such that their x-axes are parallel to the freestream direction.
- c) At least one subset of nodes must be defined by the SUBSET DEFINITION data set (User's Manual; sec. 156) for interpolating the mode shapes.
- d) All nodal subsets to be used in INTERPOLATION must be processed by the VIBRATION processor.

5.8.2 Modal Interpolation

Several methods are available for obtaining modal displacements at a general set of points. The methods can be cast into two groups: those that interpolate existing modes of vibration and those that generate displacements directly. Both groups are provided as options by the INTERPOLATION processor (User's Manual; sec. 232).

The INTERPOLATION processor performs the first stage of the interpolation process. It uses nodal subsets to define the input points and modal displacements, and the execution-control parameters as required for the selected interpolation method. It is not until later that the output points are defined and the interpolation process is completed. In the case of interpolation

to aerodynamic control points, the process is completed automatically during execution of the selected ATLAS aerodynamics processor.

No single method is suitable for performing all types of interpolation. ATLAS provides four different methods, each identified by a unique key-word in the EXECUTE INTERPOLATION control statement.

- a) Surface Spline (SURFSPLINE) -- The interpolation function is based on the small deflection bending equation of a circular, uniform-thickness, thin plate of infinite radius, simply supported along its periphery. This method is best suited when the input points are coplanar and densely distributed over the region of interpolation, and when little extrapolation is involved. Only the displacements normal to the plane defined by the input points are used.
- b) Cubic Spline Over a Lattice of Curved Beams (BEAMSPLINE)
A lattice of coplanar beams is defined by placing cubic curves through the nodes in two or more subsets, after the nodes have been arranged in order of either increasing y or increasing z. A cubic curve is then fitted through the displacements defined on each beam, so that displacement as a function of distance along each beam is defined. If extrapolation is requested, linear extrapolation of both the beam and the displacement curve is performed. For each output point, a line is defined parallel to the x-axis through the output point (or its projection onto the plane defined by the input points). A cubic curve is placed through the displacements as they are defined at the intersection of this line and each beam.

This method is best suited when the distribution of input points is sparse, or when substantial extrapolation is required. The results are dependent on the beams selected and the extrapolation options chosen. It is best to choose beams that are as straight as possible and which do not intersect, except at a common node. Only the displacements normal to the plane defined by the input points are used.

- c) Cubic Spline (MOTIONAXIS) -- This method resembles the BEAMSPLINE method except that a single beam is defined and all the input points need not lie on the beam. Extrapolation to and from the beam is linear, in a direction that the user can control through the ANGLES parameter. MOTIONAXIS is useful primarily when the

input points are almost collinear. The displacements normal to the surface defined by the input points and the x-axis are used. Optionally, the in-plane rotational displacements can also be used.

- d) Linear Extrapolation (MOTIONPT) -- The displacements at a general point are obtained by linear extrapolation of the displacements at a single point.

All of the interpolation methods require that nodal subsets be defined by the SUBSET DEFINITION preprocessor, and that those subsets be processed by the VIBRATION processor to obtain modal subsets. Those nodes included in the nodal subsets should be selected carefully since they will affect the interpolation results.

Two methods of generating displacements are provided by the INTERPOLATION processor.

- a) Surface Polynomials (POLYNOMIAL) -- The coefficients of fifth-order polynomials in the two in-plane coordinates are specified. No structural data of any kind are required if this option is used.
- b) Single Freedom Displacements (AIC) -- This method enables AIC matrices to be generated by any of the ATLAS aerodynamics processors. Each generated mode shape consists of a unit displacement of a single retained freedom. Each execution of INTERPOLATION involves two nodal subsets; one to define the nodal freedoms for which "modes" are to be generated, the other to define which freedoms are to be subjected to a unit displacement. All the nodes in the second subset must be included in the first subset. The VIBRATION processor need not be executed prior to the INTERPOLATION processor when this option is used.

The AIC key-word may be used in conjunction with any of the interpolation methods described above. When the second stage of interpolation is performed, the unit displacement of a freedom results in displacements at all output points.

5.8.3 Oscillatory Aerodynamic Forces

Calculation of the aerodynamic forces needed to complete the equations of motion for flutter and subcritical response analyses requires that the displacements and the geometry of the aerodynamic surfaces be defined. The displacements are prepared by the INTERPOLATION processor using the appropriate data from

the preceding structural analysis; the geometry is defined by the ATLAS aerodynamic input data which are independent of (though related to, of course) the geometry data used in the structural analysis. The input data are also used to specify which interpolated mode shapes are associated with each aerodynamic surface, and to define various modifications to the theoretically derived aerodynamic results.

Four different aerodynamic theories are provided by ATLAS, each particularly suited to specific types of problems.

- a) The strip-theory processor, AF1, is best suited for high aspect-ratio surfaces in subsonic incompressible flow, where the effects of chordwise flexibility are not significant. Its primary advantage is that it is inexpensive to use; its disadvantage is that it performs satisfactorily on only a limited set of aerodynamic configurations.
- b) The doublet-lattice theory processor, DUBLAT, can be applied to general multi-surface configurations in subsonic compressible flow, and can include the effects of bodies of revolution. Its primary advantage is that it can be applied to a wide variety of problems; its disadvantage is its cost.
- c) The pressure-modes theory processor, RH03, can be applied to a single main surface with optional trailing edge control surfaces in subsonic compressible flow. Its primary advantages are its precise treatment of singularities occurring at planform discontinuities and control surface singularities, as well as its very accurate prediction of pressure distributions. Its disadvantages are high cost, its inability to account for multi-surface interference effects, and its sensitivity to the location of integration (control) points.
- d) The Mach-Box theory processor, MACHBOX, calculates the aerodynamic forces for Mach numbers greater than one. It is expensive and exhibits poor accuracy.

Each of the foregoing theories is based on the following set of common assumptions that must be considered when preparing the ATLAS input data and when interpreting the results.

- a) The stream flow and analysis-frame x-axis are in the GLOBAL X-direction.

- b) Either the analysis-frame y-axis or z-axis lies in the plane of the surface.
- c) The modal displacements are in the direction normal to the surface.

Since, in practice, these assumptions can be only approximately satisfied due to camber, curvature and dihedral, it is the user's responsibility to ensure that they are sufficiently well satisfied for purposes of the analysis.

For aerodynamics analysis, the direction of the outward normal to a surface is established by the order in which the geometry data for the aerodynamic surface are defined. As illustrated in figure 5-16, the positive normal to a surface remains constant as the line through (Y_2, Z_2) parallel to the x-axis is moved on a path encircling point (Y_1, Z_1) .

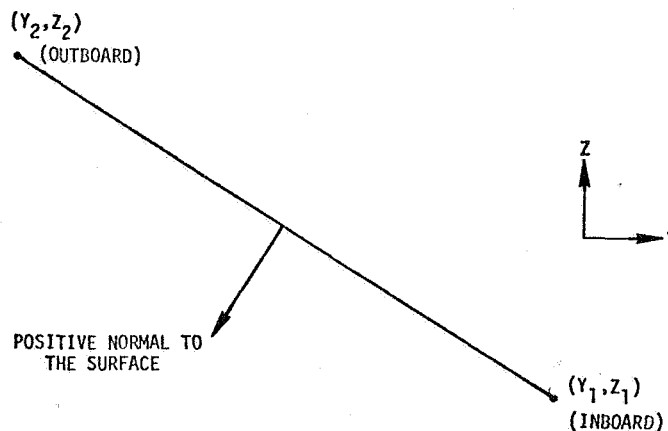


Figure 5-16. The Normal to an Aerodynamic Surface

If the geometry of the aerodynamic model is symmetric about the GLOBAL X-Z plane and the vibration modes are available for half of the structure, it is only necessary to define the aerodynamic surfaces for that half of the structure. The symmetry of the mathematical model is specified in the ATLAS execution-control statement used to execute the selected aerodynamics processor.

The unique features of each ATLAS aerodynamics processor are described in the following sections.

5.8.3.1 AF1 (User's Manual; secs. 104 and 204)

The lifting surfaces are represented as a set of strips whose edges are parallel to the GLOBAL X-axis. The strips are defined on a surface-by-surface basis by specifying the leading edge and trailing edge of the surface and the location of any control surfaces and tabs. The lift at the quarter chord due to motion at the three quarter chord of each strip is obtained using Theodorsen's coefficients (ref. 5-6). Three-dimensional effects are accommodated by using a static induction matrix. Thus, a set of generalized airforce matrices $Q(k)$ for a set of reduced frequencies, k , are calculated using

$$[Q] = \frac{1}{b_r}[Q_1] + \frac{ik}{b_r^2}[Q_2] - \frac{k^2}{b_r^3}[Q_3] + \frac{1}{b_r}[Q_4(k)] - \frac{ik}{b_r^2}[Q_5(k)]$$

where b_r = reference length

k = reduced frequency based upon b_r

$[Q_\ell]$, $\ell = 1, 2, 3$ are "component matrices" representing the noncirculatory contributions to $[Q]$

$[Q_\ell]$, $\ell = 4, 5$ represent the circulatory contributions to $[Q]$

$$[Q_\ell] = \begin{cases} [\phi]^T [dy] [S'] [D_\ell] [\phi], & \ell = 1, 2, 3 \\ [\phi]^T [dy] [S] [C(k_j)] [D_\ell] [\phi], & \ell = 4, 5 \end{cases}$$

where $[\phi]$ = matrix of vibration modes

$[dy]$ = diagonal matrix of strip widths

$[S']$ = diagonal matrix formed by multiplying the local quarter chord by the cosine of the local quarter chord sweep angle

$[S]$ = static induction matrix

$[D]$ = matrix of oscillatory derivatives

$$[C(k_j)]_{ij} = \begin{cases} \frac{J_1(J_1+Y_0)+Y_1(Y_1+J_0)-i(Y_1Y_0+J_1J_0)}{(J_1+Y_0)^2+(Y_1-J_0)^2}, & i=j \\ 0, & i \neq j \end{cases}$$

where $J(k_j)$ and $Y(k_j)$ are Bessel functions of the first and second kind and $k_j = b_j k / b_r$

Each of the foregoing matrices is partitioned on the following bases:

$$[X] = \begin{bmatrix} X_{hh} & X_{h\alpha} & X_{h\beta} & X_{hy} \\ X_{\alpha h} & X_{\alpha\alpha} & X_{\alpha\beta} & X_{\alpha y} \\ X_{\beta h} & X_{\beta\alpha} & X_{\beta\beta} & X_{\beta y} \\ X_{yh} & X_{y\alpha} & X_{y\beta} & X_{yy} \end{bmatrix}$$

where h corresponds to translation of the reference axis normal to the surface

α corresponds to the rotation of the surface about the rotation-axis component that is normal to the stream flow and which passes through the intersection of the reference axis and strip centerline.

β corresponds to the rotation of a control surface about the hinge-line component that is normal to the stream flow and which passes through the intersection of the hinge line and the strip centerline.

y is as β , but for a tab surface.

Thus, $[\phi]$ is of the form

$$[\phi] = \begin{bmatrix} \phi_{1jh} \\ \phi_{2jh} \\ \vdots \\ \phi_{1j\alpha} \\ \vdots \\ \phi_{1j\beta} \\ \vdots \\ \phi_{1jy} \\ \vdots \end{bmatrix}$$

where ϕ_y represents the appropriate displacement of the i -th strip in mode j .

Furthermore,

$$dy_{ijhh} = dy_{ij\alpha\alpha} = dy_{ij\beta\beta} = dy_{ijyy} = \begin{cases} \text{width of strip } i, & i = j \\ 0, & i \neq j \end{cases}$$

$$S'_{ijhh} = S'_{ij\alpha\alpha} = S'_{ij\beta\beta} = S'_{ijyy} = S_{ij\beta\beta} = S_{ijyy} = \begin{cases} \frac{C_i}{4} \cos \Lambda_i, & i = j \\ 0, & i \neq j \end{cases}$$

where C_i = the chord at the centerline of strip i
 Λ_i = the sweep of the quarter chord of strip i
 $dm_i = \frac{C_i}{b_r} \frac{dC_m}{dC_L}$ about the motion axis, $= (1/2 + a_i)b_i / b_r$
 (i.e., the distance from the quarter chord to the motion axis)
 $S_{ijhh} = S_{ij\alpha\alpha}$ = the "i-j" term in the static induction matrix defined in reference 5-7.
 $S_{ij\alpha h} = dm_i \cdot S_{ijhh}$

All other partitions in $[S]$, $[S']$ and $[d]$ are zero.

Each partition in $[D_\ell]$ is diagonal and is a function of strip geometry, based on the following quantities expressed as fractions of local semichord, positive in the aft direction

$$[D_1] = \begin{bmatrix} 0.0 & 0.0 & 0.0 & 0.0 \\ 0.0 & 0.0 & -4b(T_{18} + \ell T_{22}) & -4b(\bar{T}_{18} + m\bar{T}_{22}) \\ 0.0 & 0.0 & \frac{-4b}{\pi}(T_{18} + \ell T_{26} + \ell^2 T_{28}) & \frac{-4b}{\pi}(Y_9 + \ell Y_{11} + mY_{13} + \ell mY_{15}) \\ 0.0 & 0.0 & \frac{-4b}{\pi}(Y_{17} + mY_{19} + \ell Y_{21} + \ell mY_{23}) & \frac{-4b}{\pi}(\bar{T}_{18} + m\bar{T}_{26} + m^2 \bar{T}_{28}) \end{bmatrix}$$

$$[D_2] = \begin{bmatrix} 0.0 & 4\pi b b_r & -4b b_r (T_4 - 2\ell \sqrt{1-c^2}) & -4b b_r (\bar{T}_4 - 2m \sqrt{1-d^2}) \\ 0.0 & -4\pi b^2 (a - \frac{1}{2}) & -4b^2 (T_{16} + \ell T_{29}) & -4b^2 (\bar{T}_{16} + m\bar{T}_{29}) \\ 0.0 & -4b^2 (T_{17} + \ell T_{25}) & \frac{-4b^2}{\pi} (T_{19} + \ell T_{27} + \ell^2 T_{29}) & \frac{-4b^2}{\pi} (Y_{10} + \ell Y_{12} + mY_{14} + \ell mY_{16}) \\ 0.0 & -4b^2 (\bar{T}_{17} + m\bar{T}_{25}) & \frac{-4b^2}{\pi} (Y_{18} + mY_{20} + \ell Y_{22} + \ell mY_{24}) & \frac{-4b^2}{\pi} (\bar{T}_{19} + m\bar{T}_{27} + m\bar{T}_{29}) \end{bmatrix}$$

$$[D_3] = \begin{bmatrix} -4\pi b b_r & -4\pi b^2 b_r a & -4b^2 b_r (T_1 - 2T_4) & -4b^2 b_r (\bar{T}_1 - m\bar{T}_4) \\ -4\pi a b^2 & -4\pi b^3 (1/8 + a^2) & -4b^3 (2T_{13} + 2T_{24}) & -4b^3 (2\bar{T}_{13} + m\bar{T}_{24}) \\ -4b^2 (T_1 - 2T_4) & -4b^3 (2T_{13} + 2T_{24}) & \frac{4b^3}{\pi} (T_3 - 2\ell T_2 + \ell^2 T_5) & \frac{4b^3}{\pi} (Y_6 - mY_4 - \ell Y_3 + \ell m Y_1) \\ -4b^2 (\bar{T}_1 - m\bar{T}_4) & -4b^3 (2\bar{T}_{13} + m\bar{T}_{24}) & \frac{4b^3}{\pi} (Y_6 - mY_4 - \ell Y_3 + \ell m Y_1) & \frac{4b^3}{\pi} (\bar{T}_3 - 2m\bar{T}_2 + m^2 \bar{T}_5) \end{bmatrix}$$

$$[D_4] = \begin{bmatrix} 0.0 & 4b_r m_0 & \frac{4b_r m_0}{\pi} (T_{10} - 2T_{21}) & \frac{4b_r m_0}{\pi} (\bar{T}_{10} - m\bar{T}_{21}) \\ 0.0 & 0.0 & 0.0 & 0.0 \\ 0.0 & -4b(T_{12} - 2\ell T_{20}) & -\frac{4b}{\pi} (T_{12} - 2\ell T_{20})(T_{10} - 2T_{21}) & -\frac{4b}{\pi} (T_{12} - 2\ell T_{20})(\bar{T}_{10} - m\bar{T}_{21}) \\ 0.0 & -4b(\bar{T}_{12} - 2m\bar{T}_{20}) & -\frac{4b}{\pi} (\bar{T}_{12} - 2m\bar{T}_{20})(T_{10} - 2T_{21}) & -\frac{4b}{\pi} (\bar{T}_{12} - 2m\bar{T}_{20})(\bar{T}_{10} - m\bar{T}_{21}) \end{bmatrix}$$

$$[D_5] = \begin{bmatrix} -4b_r m_0 & 4bb_r m_0 (\frac{1}{2} - a) & \frac{2bb_r m_0 (T_{11} - 2\ell T_{10})}{\pi} & \frac{2bb_r m_0 (\bar{T}_{11} - 2m\bar{T}_{10})}{\pi} \\ 0.0 & 0.0 & 0.0 & 0.0 \\ 4b(T_{12} - 2\ell T_{20}) & -4b^2 (T_{12} - 2\ell T_{20})(\frac{1}{2} - a) & -\frac{2b^2}{\pi} (T_{12} - 2\ell T_{20})(T_{11} - 2\ell T_{10}) & -\frac{2b^2}{\pi} (T_{12} - 2\ell T_{20})(\bar{T}_{11} - 2m\bar{T}_{10}) \\ 4b(\bar{T}_{12} - 2m\bar{T}_{20}) & -4b^2 (\bar{T}_{12} - 2m\bar{T}_{20})(\frac{1}{2} - a) & -\frac{2b^2}{\pi} (\bar{T}_{12} - 2m\bar{T}_{20})(T_{11} - 2\ell T_{10}) & -\frac{2b^2}{\pi} (\bar{T}_{12} - 2m\bar{T}_{20})(\bar{T}_{11} - 2m\bar{T}_{10}) \end{bmatrix}$$

- a = distance from the midchord to the motion axis
 c = distance from the midchord to the control surface hingeline
 d = distance from the midchord to the tab hinge-line
 ℓ = distance from the control surface leading edge to the control surface hingeline
 m = distance from the tab leading edge to the tab hingeline
 $T_K, K = 1, \dots, 29$, are the Theodorsen control surface T-functions as presented in reference 5-8, except that T_7 is obtained using the expression in ref. 5-6 (\bar{T} denotes a tab T-function)
 $Y_K, K = 1, \dots, 24$, are the Theodorsen Y-functions as defined in reference 5-8.

The term m_0 is the lift curve slope whose theoretical value is 2π . In the expression for $[D_i]$ above, it is implied that the matrix partitions are diagonal, with an entry for each strip defined in the analysis.

Empirical Modifications

It is possible to modify the analysis performed by AF1 by substituting experimental data for calculated values in the following different ways:

- a) The terms in D_4 and D_5 may be replaced or scaled
- b) The " m_0 " distribution may be changed directly
- c) Experimental, steady-state lift distributions may be used to modify the theoretically derived " m_0 "
- d) Experimental moment distributions may be used to modify the location of the aerodynamic center

Options (a) and (b) require no further explanation, whereas options (c) and (d) are based on the premise that a wind tunnel test has been performed from which $cC_{L\alpha} \cdot \alpha$ is available.

The theoretically derived lift distribution at $k = 0$ is

$$\{l\} = 2\rho V^2 [S] [m_0] \{\alpha\} \equiv 2\rho V^2 [S] [\alpha] \{m_0\}$$

Substituting for the experimental lift distribution

$$\{cC_{L\alpha} \cdot \alpha\} = 4 [S] [\alpha]$$

where $[S]$ is nonsingular since $[S] = [S_1]^{-1}$, and where $[S_1]$ represents the downwash matrix. If α for any strip is equal to zero, the m_0 for that strip is indeterminate (and remains unchanged at 2π). Otherwise, m_0 is calculated as

$$\{m_0\} = 1/4 [\alpha^{-1}] [S]^{-1} \{cC_{L\alpha} \cdot \alpha\}$$

An example is used to illustrate how this substitution is applied in a complex case. Consider the aerodynamic idealization of an airplane with two engines illustrated in figure 5-17. Suppose lift distributions on the wing and nacelles were obtained from a test in which the configuration was pitched about the y-axis.

The pitching moment distribution, cdC/dC (where C is measured about the quarter chord) may be input directly to AF1. These data are used to relocate the aerodynamic center from the quarter chord to the experimentally obtained location, for calculating the oscillatory sectional airforces. The following term is used to transform data from the aerodynamic center to the motion axis

$$\left(\frac{\partial C_{M_i}}{\partial C_{L_i}} - \left(\frac{1}{2} + a_i \right) \right) \frac{b_i}{b_r}$$

The modification options should be used cautiously since they can easily disguise defects in the aerodynamic model and the selected method of solution. If the modifications are large or contorted, or if they change the analysis results significantly, the original modeling technique should be modified accordingly.

5.8.3.2 DUBLAT (User's Manual; secs. 116 and 216)

The lifting surfaces are represented as a set of trapezoidal boxes with two edges parallel to the x-direction. The boxes are defined on a panel-by-panel basis, where each panel is a trapezoidal part of a lifting surface. The lift at the quarter chord due to motion at the three quarter chord of each box is obtained using the doublet lattice theory described in reference 5-9. The oscillatory lift on bodies of revolution may be included, and is based on lifting body theory. Interference effects are accommodated by "interference panels" that introduce additional boundary conditions (i.e., there is no flow through an interference panel). Thus, a set of generalized airforces, $Q(k,M)$ for a set of reduced frequencies, k , and Mach number M are calculated using

$$Q(k,M) = \sum_{i=1}^n [f]_i [C_p(k,M)]_i A_i$$

where n = number of boxes
 C_p = pressure difference
 f = normal displacement
 A_i = area of box i

For a problem involving lifting surfaces only, $C_p(k, M)$ satisfies the expression $[w(k, M)] = [D(k, M)] * [C_p(k, M)]$ where $[w]$ represents the matrix of downwashes, and where a typical element of $[D]$ is

$$D_{nm} = \int_{A_m} \frac{e^{\frac{ik}{b} x_0}}{r^2} (K_1 T_1 + K_2 T_2) d\sigma$$

where

$$K_1 = I_1 + \frac{Mr}{R} \left(e^{ik_1 u_1} / (1 + u_1^2)^{\frac{1}{2}} \right)$$

$$K_2 = -3I_2 - ik_1 M^2 r^2 e^{ik_1 u_1} / R^2 (1 + u_1^2)^{\frac{1}{2}} \\ - \frac{Mr}{R} \left[(1 + u_1^2) \frac{\beta^2 r^2}{R^2} + 2 + \frac{Mr u_1}{R} \right] \frac{e^{ik_1 u_1}}{(1 + u_1^2)^{\frac{3}{2}}}$$

$$u_1 = (MR - x_0) / \beta^2 r$$

$$k_1 = \omega r / u_\infty$$

$$\beta = \sqrt{1 - M^2}$$

$$R = \sqrt{(x_0^2 + \beta^2 r^2)}$$

$$I_1 = \int_{u_1}^{\infty} \frac{e^{ik_1 u}}{(1 + u^2)^{\frac{3}{2}}} du$$

$$I_2 = \int_{u_1}^{\infty} \frac{e^{ik_1 u}}{(1 + u^2)^{\frac{5}{2}}} du$$

and where x_0, y_0, z_0 , are the x, y , and z distances from a point in the sending box, m , to the receiving point, n .

$$r = \sqrt{y_0^2 + z_0^2}$$

$$T_1 = \cos (y_n - y_m)$$

$$T_2 = (Z_0 \cos Y_n - y_0 \sin Y_n)(Z_0 \cos Y_m - y_0 \sin Y_m) / r^2$$

$$Y_i = \text{dihedral of box } i$$

References 5-9 and 5-10 contain the derivation of these equations, as well as the equations governing slender body theory and interference effects.

Empirical Modifications

The equations presented above do not account for the thickness of the lifting surfaces. This thickness causes the local velocity at a box to be different from the freestream velocity. The difference is called the "velocity profile" which is a function of the planform shape. If the velocity profile is known, it can be used in DUBLAT to modify the freestream velocity used in deriving the generalized airforces.

The pressure distributions generated by DUBLAT can be modified either by scaling the calculated pressures or by replacing them. If the rate of change of pressure measured empirically is different from that calculated by DUBLAT for zero reduced frequency, scale factors can be deduced that will "correct" the calculated pressures for all reduced frequencies.

These modification options should be used cautiously since they can easily disguise defects in the modeling techniques. If the modifications are large or contorted, or if they change the analysis results significantly, the original modeling technique should be investigated and revised.

Saving Quasi-Inverse Matrices

A large part of the cost of DUBLAT is spent in calculating and "inverting" the matrix [D] shown in the foregoing equations. This matrix is a function of reduced frequency, Mach number, box distribution and symmetry, but not of the modal data. The "inverted" [D] matrices, known as quasi-inverse matrices, may be saved for reuse.

5.8.3.3 MACHBOX (User's Manual; secs. 136 and 236)

The following types of aerodynamic models can be analyzed by the MACHBOX supersonic oscillatory aerodynamics processor

- a) A single planar surface (wing) with up to 45° dihedral measured from the GLOBAL X-Y plane
- b) Two planar surfaces (wing and tail) with vertical separation and separate dihedrals of up to 45° placed so that only one surface (tail) is influenced by the other (wing)
- c) As b) except the influenced surface (tail) lies in the GLOBAL X-Z plane at Y = 0

A full account of the theoretical basis for MACHBOX is presented in reference 5-11. Basically, the lifting surfaces are represented as a set of uniform rectangular boxes whose chordwise dimensions are M^2-1 times their spanwise dimensions. A set of generalized airforce matrices $Q(k,M)$ is calculated for a set of reduced frequencies, k , using

$$\begin{aligned} [Q(k,M)] &= \sum_{i=1}^n \iint_{A_i} [f(x,y)]_i [C_p(x,y)]_i \, dx dy \\ &= \sum_{i=1}^n \iint_{A_i} [f(x,y)]_i \left[\frac{D\phi}{Dt} (x,y) \right]_i \, dx dy \end{aligned}$$

where A_i = area of box i
 n = number of boxes
 f = normal displacement
 C_p = oscillatory pressure difference
and ϕ = velocity potential as related to the upwash, w , through:

$$\phi_i = -\frac{1}{\pi} \iint_S \frac{W(x,y) e^{i\omega x_0/a\beta^2} \cos(\omega R_h/a\beta^2)}{R_h} \, dx dy$$

where S denotes the whole surface, and

x_0, y_0, z_0 are the $x, y,$ and z distances from a point on the surface to the receiving point at the center of box i

a = speed of sound

$$\beta = \sqrt{M^2 - 1}$$

ω = circular frequency of oscillation

$$R_h = \sqrt{x_0^2 - \beta^2 y_0^2 + z_0^2}$$

By assuming the downwash is constant over each box, the velocity potential can be expressed as

$$\phi_i = \sum_{j=1}^n w_j C_{ij}$$

where the velocity-potential influence coefficient C_{ij} is a function of the Mach number, reduced frequency and the relative location and orientation of boxes i and j . C_{ij} need be calculated only once, but may be used for any pair of boxes with the same physical orientation. The matrices of C_{ij} can be saved for reuse in subsequent analyses.

The major drawbacks of the Mach-Box approach are its high cost, and the large fluctuation in pressure distributions that result from imposing a grid of rectangular boxes on a surface that has swept leading or trailing edges. The magnitude of the fluctuations is not generally affected by the fineness of the box grid, though their effect on the reliability of generalized airforces is usually lessened by selecting finer grids. The fluctuations are least malign when the Mach number is such that the Mach boxes intersect the leading edge in a regular pattern, and the trailing edge effects are best represented when box centers lie just ahead of the trailing edge. The number of boxes should be chosen to give as good a placement of boxes on the tail as is consistent with meeting the first two requirements.

It is possible to reduce the cost of performing an analysis on configurations with small dihedral angles by using the options DIH1 and/or DIH2. These key-words cause the influence of the left hand surface on the right hand surface to be calculated as if the surfaces were coplanar, thereby reducing the cost of

calculation by an order of magnitude without significant loss of accuracy. At the same time, the effect of dihedral is included when calculating the influence of the wing on the tail.

The user can also trade cost against accuracy by specifying the accuracy to which the velocity potential coefficients are to be calculated (AICTOL option). Costs increase rapidly if an accuracy of less than one per cent is requested, and such gains in accuracy are probably insignificant in view of the pressure fluctuation problem.

The pressure fluctuations tend to be self canceling during calculation of the generalized airforces. However, better behaved pressure distributions can be obtained by using the subdivision option (SUBD). Broadly speaking, this option combines the effects of many sending boxes into a single receiving point, but tends to be quite expensive.

The pressure fluctuations can also be reduced relatively inexpensively by smoothing them using a least squares modeling method applied either to the whole surface or to each chordwise strip of boxes (SURFIT or CHORDFIT option). It is questionable whether there is any significant effect on the generalized forces when these modeling methods are used.

Generally, for those boxes cut by surface edges, the influence coefficients are scaled automatically by the ratio of the box area lying on the planform to the total area of the box. This scaling can be inhibited by use of the option PLYWOOD. The effect is equivalent to using a planform outline that is defined by all Mach boxes whose centers lie inside the planform. A measure of the uncertainty of the Mach-Box method can be ascertained by comparing the results obtained with and without this option.

5.8.3.4 RHO3 (User's Manual; secs. 150 and 250)

RHO3 can be used only for a single main surface with optional trailing edge control surfaces. No aerodynamic influence from any other surface or body can be taken into account. It is intended to provide extremely accurate pressure distributions at moderate cost, and is especially useful in cases where leading edge, side edge and control surface boundary singularities have a significant effect on the aerodynamic forces. "Singularities" that occur in the potential flow approximation generally correspond to large pressure changes in the real flow, that are often inadequately represented by box methods.

The theoretical basis for RHO3 is described in reference 5-12. The method calculates pressure distributions as linear

combinations of assumed pressure-mode functions that have been selected to represent various dominant aspects of the pressure distribution.

A set of generalized airforce matrices $[Q(k,M)]$ is calculated for a set of reduced frequencies using

$$[Q(k,M)] = \iint_S [f(x,y)][C_p(x,y,k,M)] dx dy$$

where S = the surface
 f = normal displacement
 C_p = oscillatory pressure difference.

The unknown oscillatory pressure distribution is expressed as a linear combination of known functions, $p_j(x,y)$, (the assumed pressure modes):

$$C_p(x,y) = \sum_j a_j p_j(x,y)$$

where a_j are obtained from the equations

$$\{W(x_i, y_i)\} = [C_{ij}] \{a_j\}$$

$$C_{ij} = \frac{1}{\pi} \iint_S p_j(\xi, \eta) K(x_0, y_0, k, M) dS$$

and where K = the kernel function
 S = the surface
 k = reduced frequency
 M = Mach number
 $W(x_i, y_i)$ = downwash at the i -th downwash point, and
 x_0 and y_0 are the x and y distances from the downwash point to the point represented by (ξ, η)

The quality of the integration of the pressure modes can be significantly affected by the choice of downwash points in RH03. They should be chosen so that they are not located close to areas on the planform where singularities are likely to occur, such as the periphery of the planform and control surfaces, and streamlines that pass through points at which the leading and trailing edge sweep angles change. Figure 5-18 illustrates certain areas that should be avoided.

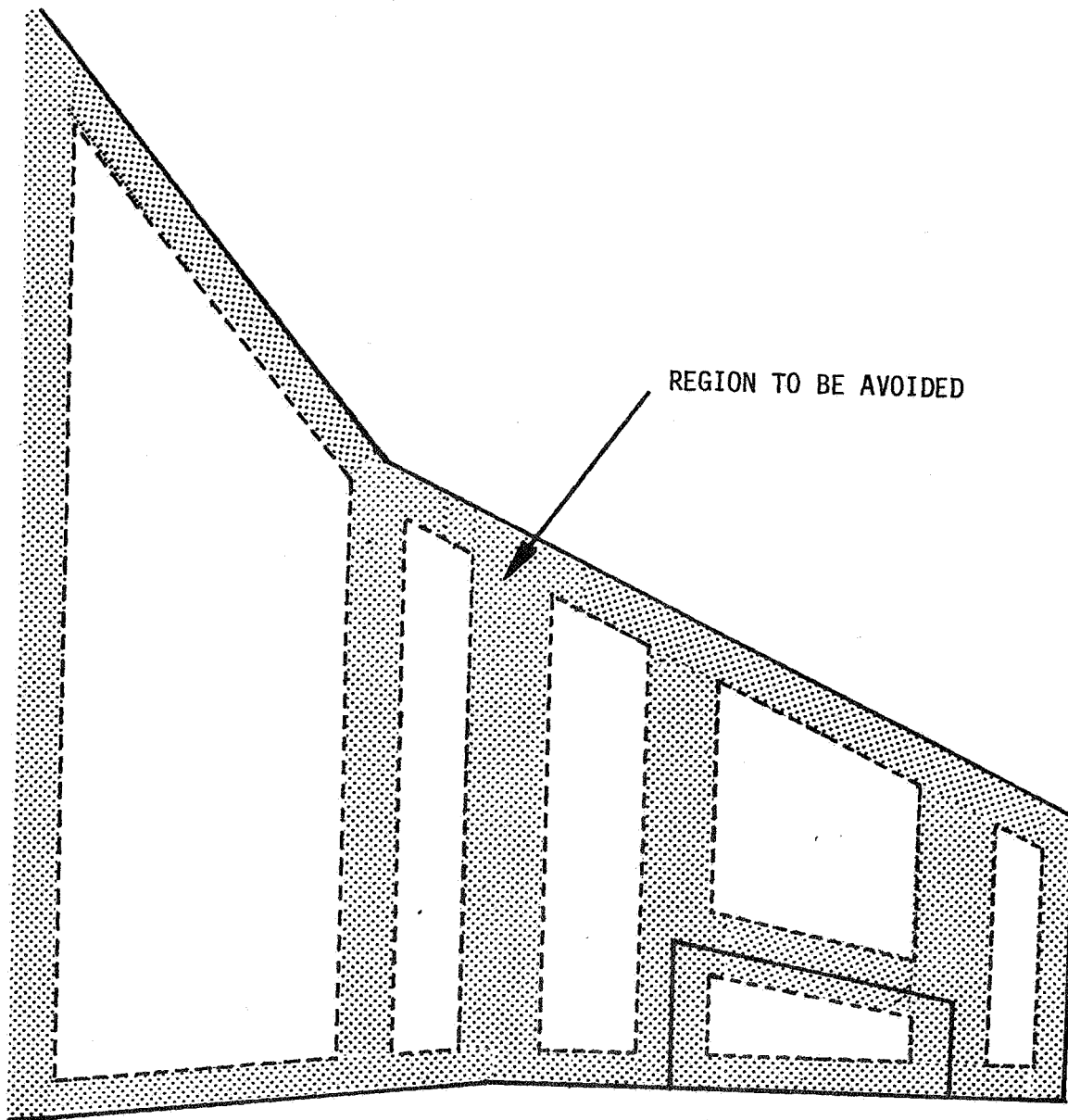


Figure 5-18. Placement of RH03 Downwash Points

Empirical Modifications

The effect of planform thickness can be accounted for through the use of the velocity profile option. This permits the local velocity at each control point to be used in place of the freestream velocity.

Saving C-Matrices

Since a substantial part of the RHO3 execution cost is in generating influence coefficient matrices that are dependent only on planform shape, downwash point location, reduced frequency and Mach number, they are saved automatically on the data file RHO3RNF. They may be used to reduce the cost of subsequent analyses in which the same aerodynamic data are used.

5.8.4 Residual Flexibility Effects (User's Manual; sec. 220)

When higher-frequency vibration modes are eliminated in order to reduce the size of the generalized mass and stiffness matrices, corresponding structural flexibilities are also eliminated. It is possible to account for that flexibility in the flutter equation by including it in the generalized airforce matrix. The FLEXAIR processor performs this function.

FLEXAIR uses the original mass and stiffness matrices and the corresponding vibration modes and generalized stiffness. It requires that the aerodynamic influence coefficients (AIC's) be available for all structural freedoms that are aerodynamically significant (i.e., those associated with lifting surfaces). From these data, FLEXAIR calculates a set of generalized airforces that resemble those from the ATLAS aerodynamics processors except that they are functions of altitude.

FLEXAIR can be used to generate generalized airforces from AIC's by setting the execution-control parameter FLUTFREQ to zero.

5.8.5 Combination and Interpolation of Airforce Matrices (User's Manual; sec. 202)

The generalized airforce coefficients approximate simple, continuous functions of reduced frequency within the range of reduced frequencies used for flutter analysis. Furthermore, aerodynamic forces at a large number of reduced frequencies are required for calculating accurate flutter speeds. Since the cost of obtaining these aerodynamic forces is high, it is usually desirable to calculate a small number (≥ 4) of them as a function of reduced frequency, and then use interpolation to obtain data

at additional reduced frequencies. The ADDINT processor performs this function.

The ADDINT processor can also be used to combine sets of airforce matrices when, for example, generalized airforces have been obtained separately for different surfaces by using different ATLAS aerodynamic processors. The matrices associated with a certain reduced frequency are simply added. Thus, all matrices involved must be the same size and must relate to the same set of modal data. If AIC matrices are to be combined in this way, each set must include rows and columns pertaining to all the freedoms in the combined matrices.

The ADDINT processor must always be executed prior to executing the FLUTTER processor.

5.8.6 Flutter Analysis (User's Manual; secs. 122 and 222)

The purpose of a flutter analysis is to calculate the structural damping required to produce neutrally stable oscillations of a structure at a given speed. This is achieved by generating and solving an equation of motion of the form

$$\left[\omega^2 [M] - (1 + ig + ig_i) [K] + \frac{1}{2} \rho V^2 [Q] \right] \{q\} = \{0\}$$

where

$[M]$ = mass properties of the structure

$[K]$ = stiffness properties of the structure

$[Q(k, M)]$ = aerodynamic forces resulting from oscillations of the structure

ω = circular frequency of oscillation

g = fictitious structural damping (positive or negative) required to produce constant amplitude oscillation

g_i = actual structural damping existing in each coordinate

ρ = freestream density

V = freestream velocity

k = reduced frequency of oscillation ($b\omega/V$)

b = reference length

The FLUTTER processor creates the flutter equation of motion using previously-calculated User Matrices representing (generalized) mass and stiffness, and the (generalized) airforces generated by the ADDINT processor.

The equation of motion can be modified by changing individual elements in the mass and stiffness matrices, by adding structural damping coefficients, and by selecting a subset of the rows and columns in the input matrices.

The density, ρ , can be specified in several ways. If the airforce matrices include the effects of residual flexibility, the altitude for which they were included should be used in the flutter analysis. Otherwise, the generalized airforces are independent of altitude so that different altitudes can be investigated, or a search for a "matched-point" solution can be performed. In the latter case, FLUTTER performs a series of flutter solutions, changing the altitude for each one to find the altitude at which the lowest flutter speed equals the freestream velocity. In all these cases, the density is calculated using standard atmosphere equations. If the standard atmosphere equations are not appropriate, the density can be specified directly.

FLUTTER solves the equation of motion to obtain values of the required structural damping and oscillatory frequency for all vibration modes and for each input reduced frequency using Laguerre iteration as described in reference 5-13. This method permits the eigenvalues of the flutter equation to be calculated in the same order at each reduced frequency, thereby permitting flutter speeds to be determined and plots of "velocity vs. damping" and "velocity vs. frequency" to be generated. Linear interpolation is used to obtain flutter speeds (i.e, speeds at which the fictitious structural damping required for neutral stability equals zero, or up to three user specified values). Eigenvalues and adjoint eigenvectors may be obtained at these flutter speeds and at selected reduced frequencies. V-g and V-f plots can be created by using the GRAPHICS postprocessors (see sec. 6.3).

It is possible to affect the cost and accuracy of the flutter solution in several ways. The cost of executing FLUTTER is linearly dependent on the number of reduced frequencies used. However, if the difference between two adjacent reduced frequencies is too large, the iteration process may fail to calculate the eigenvalues in a consistent order, and spurious flutter speeds may result. The accuracy at each reduced frequency can be improved by increasing the number of iterations used to calculate them at each reduced frequency. In practice, one iteration is usually sufficient (the method automatically

uses several iterations to find the solution at the first reduced frequency, and thereafter each new solution uses the latest solution as a starting point). It is preferable to increase the number of reduced frequencies rather than the number of iterations.

The cost can also be reduced by specifying a "window" to limit the range of velocities and frequencies at which flutter speeds are generated. The solution is performed only at the reduced frequencies which are within the range defined by upper and lower limits on the velocity and frequency. The "window" feature can also be used to avoid calculation of results which are of no interest, and to increase the reliability of the matched point search when many vibration modes experience instabilities.

5.9 SUBSTRUCTURED MODELS (User's Manual; secs. 130 and 230)

Stress analyses and dynamic analyses can be performed using ATLAS substructured models. Substructuring is the procedure by which the total structure is divided into separate parts, each of which is idealized separately in the same manner as described previously for an ATLAS analysis without substructuring. Each part is identified by a substructure number. When all of the component substructures are interacted (joined together), they form the total structural model. Merging the data for the substructures and performing the required solutions are performed most conveniently by using the ATLAS execution-control procedures described in section 6.

Use of a substructured analysis approach is recommended when one or both of the following conditions exist.

- a) The problem size is relatively large
- b) Parametric studies or successive modifications of a relatively small and well defined part of the total model are required.

An ATLAS substructured analysis can be represented schematically by a singly-connected tree diagram as shown in figure 5-19. Each node in the diagram represents a substructure, and each line represents how the substructures interact. A maximum of 28 substructures can be interacted simultaneously in forming the next higher-level substructure. There is no limit on the number of levels used to define the interact tree. A particular analysis, however, is limited to a maximum of 999 substructures.

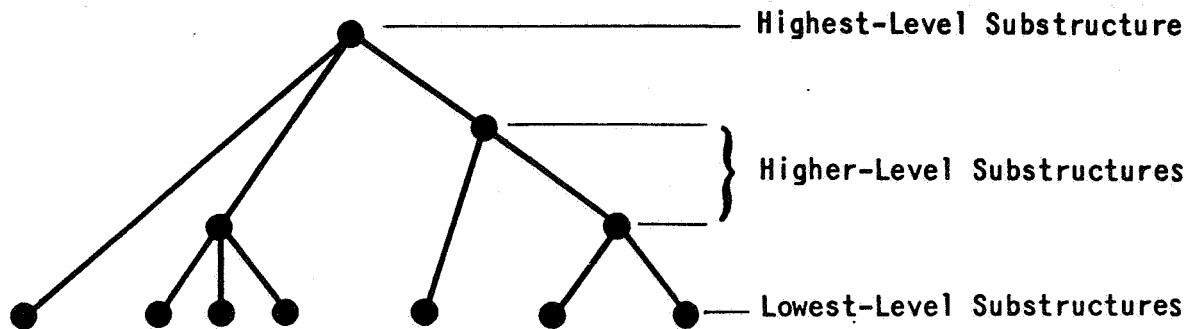


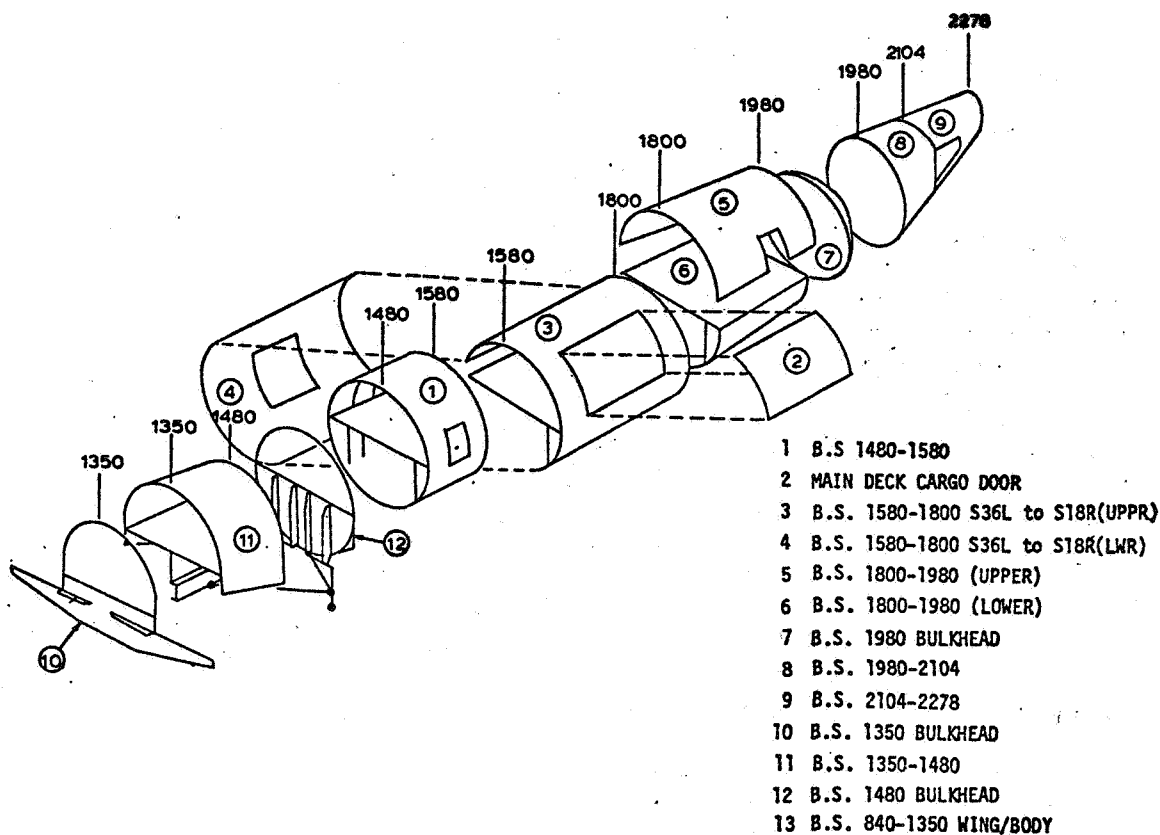
Figure 5-19. A Substructured Model Interact Diagram

The basic substructures for which the user supplies all structural, loads, mass, and boundary condition data are referenced as the "lowest level" substructures. These basic data are defined in terms of SETs and STAGEs in the usual manner (see secs. 5.2, 5.3 and 5.4). Each lowest level substructure is defined as (equivalenced to) a SET and STAGE in the INTERACT data set. The size limitation for each substructure is, therefore, the same as for a SET; viz. 4095 nodes and 32 767 elements. When defining an interact tree, two or more of the lowest level substructures are merged to form the next higher level substructure. This process is continued until all of the substructure components are merged into a single substructure that is referenced as the highest level substructure. An example substructured model is illustrated in figure 5-20. Example problems are also presented in sections 8.6 and 8.7.

Detailed descriptions of the ATLAS substructure INTERACT data and analysis procedures are presented in section 130 of the User's Manual.

5.9.1 Substructured Stress Analysis

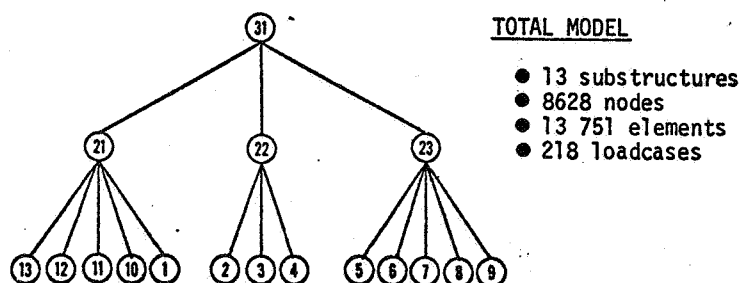
For a substructured stress analysis, the elemental stiffnesses and nodal loads are merged at the lowest level. These matrices are reduced according to the specified INTERACT data, and are then merged to form the next higher level substructure. Reduced matrices for a substructure correspond to the "retained" freedoms for that substructure, as specified by the boundary condition data. The higher-level substructures are



(a) Substructure Arrangement

SUBSTRUCTURE		1	2	3	4	5	6	7	8	9	10	11	12	13	21	22	23	31
NODES		854	226	1233	813	944	1133	186	562	667	377	1103	329	241	258	185	184	124
ELEMENTS		1156	625	1545	1146	1471	1734	382	1060	1036	726	2152	718	(KRED)	0	0	0	0
LOADCASES		36	12	94	27	15	50	1	4	8	12	7	24	0	71	110	61	218
FREEDOMS	FREE	1331	574	1831	1422	1210	1389	286	1090	1387	1073	2148	826	474	555	219	402	378
	RETAINED	382	97	446	291	273	270	166	310	148	185	458	259	247	190	378	188	0
	SUPPORTED	60	13	94	41	68	50	116	349	487	132	38	4	725	0	0	0	0

(b) Details of Substructured Model



(c) Substructure Interact Diagram

Figure 5-20. Example Substructured Model

represented strictly by their nodes and reduced matrices. No structural elements or loads are associated explicitly with them.

Those data matrices required for back-substitution (displacement calculations) are generated during the reduction process for each substructure. The merge-reduction process is continued until the highest substructure is reached. The resulting interaction equations for the highest substructure are then solved to determine its interfaced nodal displacements for all loadcases. These interface displacements are then partitioned into matrices associated with the retained freedoms for each of the substructures that interact in forming the highest substructure. Back-substitution is performed using these partitioned displacement matrices to obtain a total set of nodal displacements for the component substructures. This partition and back-substitution process is repeated until all lowest level substructures are reached. The STRESS processor can then be executed for each of the lowest level substructures to calculate element stresses.

5.9.2 Substructured Vibration Analysis

A substructured vibration analysis follows a somewhat different course than that used for a stress analysis. Stiffness matrices for the substructures are merged and reduced in the same manner as for a stress analysis. Loads matrices, however, are not required for a vibration analysis. Instead, mass matrices are merged and reduced according to the selected options available at the lowest level of the interact tree. The options are as follows:

- a) Compute directly either the diagonal or the non-diagonal reduced mass matrix for a substructure by execution of the MASS processor (Mass OPTION=2 and OPTION=3 described in sec. 5.3.5).
- b) Perform a Guyan reduction on the gross mass matrix for a substructure to generate its reduced mass matrix (Mass OPTION=4).

The reduced mass matrices for the intermediate and highest level substructures are obtained through a merge and Guyan reduction process. Optionally, the reduced mass matrix for the highest substructure can be computed directly by executing the MASS processor with OPTION=2 or OPTION=3.

The merge-reduce process for each higher level substructure is continued until reduced stiffness and mass matrices are generated for the highest substructure. This substructure is identified by a structural model SET number in the INTERACT data.

Vibration modes are then calculated for this SET by executing the VIBRATION processor in the usual manner (see sec. 5.7).

5.9.3 Automated Interaction Criteria

The facility for automatically detecting and retaining those freedoms that interact between the different substructures is provided by the INTERACT preprocessor.

The basic approach used by ATLAS is that 1) whenever two nodes from different substructures lie within the user-specified distance tolerance of each other and have the same analysis frame, they are assumed to interact. The corresponding active freedoms that are either "free" or "retained" for both nodes are regarded as the interacting freedoms for those nodes. The boundary conditions for these freedoms are automatically set to RETAIN. If the UNIQUE NODE NUMBERS data record is specified in the INTERACT data, the user-assigned numbers of the interacting nodes must be identical in addition to the foregoing criteria; 2) whenever two nodes with interacting freedoms are found, they are represented by a single node in the next higher level substructure. Those freedoms in a lower level substructure that interact (and hence are retained) are assigned the FREE activity label at the next higher level.

This feature relieves the burden on the user of manually compiling lists of interacting freedoms and their substructure correspondence. It also provides an independent check of the geometry definition of substructure interfaces.

The following information is pertinent when using the automatic interaction criteria.

- a) User Specified Tolerance -- When a distance tolerance between interacting node locations is not specified, the default value is 10^{-8} units. The selected tolerance must be less than the shortest distance between any of the nodes in a substructure that are to interact. Otherwise, the wrong nodes may get interacted (lumped together) at a substructure interface.
- b) Freedom Activity -- Only the active freedoms that are either free or retained for an interacting node are actually interacted. The freedom activity information is available only after the STIFFNESS processor is executed. The INTERACT preprocessor is, therefore, designed to follow one of the following paths:
 - 1. If the STIFFNESS processor is not executed prior to reading the INTERACT data, all freedoms are assumed

to be "active" (i.e., have nonzero stiffness). All common "free" or "retained" freedoms are assumed to interact, and are tagged as "retained" freedoms.

2. If the STIFFNESS processor is executed prior to reading the INTERACT data, the proper activity of each freedom is taken into account. Only the common "free" or "retained" active freedoms are assumed to interact, and are tagged as "retained" freedoms.

Path (2) ensures that no inactive freedom is retained and thereby, it avoids encountering singularities in solving the interaction equations. This danger exists if path (1) is followed, unless the user has supported all inactive freedoms in the basic BC data for the interacting substructures. It is, therefore, recommended that path (2) be followed.

Suppression of Automatic Interaction

The automatic interaction feature can be suppressed by input of the "NO GEOMETRIC RETAINS" INTERACT data record. In this case, the program does not generate the substructure connectivity. The interacting freedoms information and the original boundary conditions, as supplied by the user, remain effective.

The user has the option of letting the program generate the geometric retains, and then overriding those data by manually supplying the desired boundary conditions using the "BC CHANGE" INTERACT data.

Treatment of Non-Interacting Retained Freedoms

Any nodes that do not interact but have one or more "retained" freedoms are included in the next higher substructure. The retained freedoms that do not interact in forming a higher level substructure remain as retained freedoms in that higher substructure.

- c) Use of "UNIQUE NODE NUMBERS" -- It is not mandatory that the user identification numbers of the interacting nodes be identical. However, the user can optionally impose this restriction by including the UNIQUE NODE NUMBERS data record in the INTERACT data set. This offers the advantage of being able to identify a node in any substructure (the total model) by its unique number. This feature is useful in specifying BC CHANGE data and in defining subsets of nodes for the SET corresponding to the highest level substructure. Node subsets, for

example, are generally required for subsequent vibration analysis of the highest level substructure.

5.9.4 Treatment of Loadcases

All loadcase labels are supplied by the LOADS data set (see sec. 5.4). Furthermore, all labels are assumed to be consistent throughout the total model. This results in the INTERACT preprocessor automatically generating the merging information for loadcases that have the same label for the different substructures. Thus, the highest level substructure has associated with it, a list of all the different loadcase labels, as specified for the total structure.

During the back-substitution phase, the displacements for selected loadcases can be calculated for selected substructures.

5.9.5 Treatment of Analysis Frames

The analysis frame identifiers, as defined by the NODAL data sets, are assumed to be consistent throughout the total structural model. Those node analysis frames specified for the lowest level substructures are used throughout substructured analysis.

5.9.6 Multi-Level Interaction

There is no inherent limitation on the number of levels in an interact tree. The only limitation is that the tree must be singly-connected; i.e., a substructure can only be interacted once. Intermediate level substructures can be interacted together with one or more lowest level substructures. Whenever a particular nodal freedom is to interact more than once in forming multiple higher level substructures, it must be retained manually for the appropriate substructure, based on the following criteria.

When two or more substructures are interacted, the resulting higher level substructure contains only those nodes that have one or more retained freedoms in its component interacting substructures. All other nodes are reduced out at the lower level. When an interacting node is detected, its interacting freedoms are set to FREE in the higher level substructure and are reduced out at that level in a subsequent reduction operation. Such freedoms are then lost as far as further interaction at a still higher level is concerned, unless they are retained manually for the higher level substructure. The user must, therefore, ensure that those freedoms which interact at multiple substructure levels are retained by appropriate use of the BC CHANGE data subset of the INTERACT data set.

5.9.7 Recommended Practices

The following practices are recommended when performing substructured analyses.

- a) Execute the STIFFNESS processor before reading the INTERACT data set.
- b) Use unique node numbers for convenient definition of node subsets, as generally required for vibration/flutter analyses of the highest level substructure.
- c) The interact distance tolerance must be less than the shortest distance between any of the nodes in a substructure that are to interact; this must be the case for each substructure.
- d) For multi-level substructured analyses, those freedoms that interact in forming a particular substructure and which interact again at a higher level, must be retained by using the BC CHANGE data subset for that particular substructure.
- e) Carefully check the printout from the INTERACT postprocessor (User's Manual; sec. 230) before proceeding with the merge-reduce operations, to verify the substructure interaction model.

5.10 STRUCTURAL RESIZING (User's Manual; secs. 112 and 212)

The ATLAS System provides structural design capabilities for automated resizing of general finite element models. Properties (areas and thicknesses) of selected finite elements are modified (resized) according to the geometric and/or margin-of-safety constraints specified by the DESIGN data set (sec. 112 in the User's Manual). Those model-definition data sets, in addition to a DESIGN data set, that are required for performing automated structural resizing are the same as that required for a basic stress analysis (see table 5.1 and sec. 5.5). The overall design capabilities are intended primarily for preliminary structural design in a large problem environment.

The four basic structural design methods provided by ATLAS are:

- a) A Fully-Stressed-Design (FSD) technique
- b) A Thermal Fully-Stressed-Design (TFSD) technique
- c) A composite-structure optimization procedure
- d) An element property smoothing function

Strength-design criteria are used by the FSD technique, the TFSD technique, and by the optimization procedure. The smoothing function is provided primarily for making non-stress-dependent changes to selected element properties. Use of this function, for example, allows interaction with the design process to alter the convergence characteristics of the automated design solutions.

The design process in ATLAS is iterative in nature, because redistribution of the internal loads due to the element resizing is determined by the "next" stress analysis. Each design iteration is identified by a user-specified CYCLE number, wherein a design cycle includes the following:

- a) Generation of the structural stiffness matrices
- b) Solution for nodal displacements and element stresses
- c) Calculation of margins of safety based on the specified design criteria
- d) Updating the element properties

The design process is influenced by the user through criteria specified by the DESIGN input data set which includes geometric and margin of safety constraints on the element properties, and through the EXECUTE DESIGN control statement parameters (sec. 212 in the User's Manual) that specify the number of design cycles (iterations) to be performed, and the convergence requirements of the design solution.

The ATLAS automated design capabilities have been developed particularly for the large problem environment with emphasis given to the "optimality approach" except for composite structure for which a "regional search" is combined with a "math-programming approach." It should be noted that use of an optimality design technique such as the FSD and TFSD procedures sometimes results in a non-converging solution, or a solution that is not the minimum weight design. However, the state-of-the-art is such that only the optimality criterion methods are economically feasible for large models. Intuitive arguments are often used in selection of the design criteria that, when satisfied, produce a minimum or "almost" minimum weight design. Application of these methods, therefore, requires that a level of experience be developed for selection of the design constraints, for judicious use of the convergence control devices as provided by ATLAS, and for interpretation of the design results.

Element stresses used by the DESIGN processor in performing the element resizing are those calculated by the STRESS processor except for the BEAM elements. When BEAM elements are included in a structural model, a PROPERTY data subset must be included in the DESIGN data set. This data subset is used in the conversion of element moments and forces to stresses. The remaining DESIGN data subsets are used, as required, to introduce practical design constraints, to guide the design solution into a desired range, or to improve the convergence characteristics of the design solution in selected regions of the model.

The following types of design constraints can be specified by the DESIGN data set.

- a) Lower Bounds -- Minimum (lower-bound) element property values
- b) Upper Bounds -- Maximum (upper-bound) element property values
- c) Fixed Data -- Selected element property values that are not to be changed during resizing
- d) Restrain Sizing -- Selected elements or regions of the structural model that are not to be changed during resizing
- e) Input Margins -- Margins of safety for selected element properties that must be satisfied during resizing

Options for updating a DESIGN data set are provided for user interaction between successive design cycles. These features, in addition to the element-property smoothing function, allow the user to constrain the design process or to influence the behavior of the iterative design solution.

The automated design process is performed by including a PERFORM DESIGN statement in the execution-control deck (see sec. 6 and appendix E in the User's Manual). The following types of convergence requirements can be used.

- a) The maximum allowable change of the total weight of the model after each design cycle; $W^* = |W(i+1) - W(i)|$
- b) The maximum allowable relative change of the total weight of the model after each design cycle; $R = |W^* / W(i+1)|$, where W^* is defined by (a).
- c) The maximum allowable difference between the "new" and the "old" property values for selected elements

Iteration on a solution is performed automatically until all specified design criteria are satisfied, or until the specified number of design cycles has been performed.

Formatted printout of the DESIGN input data, as well as the resized element properties and margins of safety, are generated by including appropriate "PRINT" statements in the execution-control deck (see sec. 6 and the User's Manual; sec. 212). The resized element properties and margins of safety can also be plotted as contours or graphs using the GRAPHICS postprocessor (see sec. 6.3).

Discussions of the various ATLAS structural design capabilities are presented in the following subsections.

<u>Section</u>	<u>Design Method</u>
5.10.1	Fully Stressed Design (FSD)
5.10.2	Thermal Fully Stressed Design (TFSD)
5.10.3	Composite Optimization
5.10.4	Element Property Smoothing

Further details of the design methods are presented in the ATLAS "Design Module Theory Document."

5.10.1 Fully Stressed Design (FSD)

The fully-stressed-design method can be used to resize the following types of stiffness finite elements.

ROD	SPAR	PLATE	SROD
BEAM	COVER	GPLATE	SPLATE

In performing an automated structural design, the resize procedures use the constraints specified by the DESIGN data set and a resize algorithm based on element-property margins of safety.

$$A^* = A(1 + MSI)/(1 + MSC)$$

where A^* = new property value
 A = old property value
 MSI = input (specified) property margin of safety
 MSC = calculated property margin of safety

Selected element properties (thicknesses and cross-sectional areas) are the design variables in these expressions. The remaining element properties are changed in the same proportions as their related design variables are modified.

Calculation of an equivalent margin of safety, MSC, is such that the corresponding new element property will be just adequate for the applied loading. That is, the applied stress will equal the corresponding allowable stress. Calculation of the margins of safety is based on the assumption that internal loads do not change when design variables are changed. In order to satisfy both strength and buckling requirements, the various calculated margins of safety are screened, and the minima thereof are used to establish a margin-of-safety design envelope prior to use of the foregoing resize algorithm. Margins can be calculated according to the following different criteria.

- a) Strength -- Hill's failure criterion (ref. 5-14) is used to satisfy the strength requirements. For the one-dimensional case, the equation is

$$\text{MSC(Strength)} = (F/f) - 1$$

where "f" is the applied stress, and "F" is the allowable stress as defined by the MATERIAL data set. For plate-like elements, the condition of transverse isotropy is assumed.

- b) Panel Buckling -- The panel buckling criterion is applied to the SPAR, COVER, PLATE, GPLATE, and SPLATE elements. By default, the allowable compression and shear buckling stresses are those defined by the MATERIAL data set. These data are used as constant buckling allowables. Optionally, property-dependent allowable buckling stresses can be specified for selected elements by input of a "TABLE" DESIGN data subset.
- c) Local Buckling -- The local buckling criterion is applied to the PLATE, GPLATE, and COVER elements only when a "MODULUS" DESIGN data subset is input. Local buckling allowables are calculated by use of the specified stress-dependent elastic moduli, and the assumptions of a rectangular isotropic plate on simple supports. Optionally, the effect of plate stiffeners in two orthogonal directions (the effective elemental "free spans") can be included in the calculation of local buckling allowables through input of a DETAIL data set (sec. 114 in the User's Manual).

Two options are provided for calculation of the buckling margins of safety when the TABLE or local-allowable stresses are element-property dependent. Either a first-order "predictor" method or a "local" iteration to convergence can be used to

account for the change in allowable stress due to a change in the design property as shown by the following expression.

$$MSC(\text{Buckling}) = [1 - R(t_0)] / [R(t_0) - 1 - t_0 R'(t_0)]$$

where $R(t_0)$ represents the buckling stress interaction expression based on the initial property t_0 , and $R'(t_0)$ represents the first-order derivative with respect to the design variable. The objective is to establish the new property t_1 such that $R(t_1) = 1$.

The default buckling interaction formula used for the plane stress case is that corresponding to the classical buckling theory of simply-supported, thin plates.

$$R = [R_1 / (1 - R_1^2)]^4 + R_2 / (1 - R_1^2)$$

where each "R" denotes the ratio of an applied stress in the 1, 2, or 1-2 material directions to the corresponding allowable stress. Optionally, different interaction equations can be specified for selected elements by an equation of the form

$$R = a * R_1^{e_1} + b * R_2^{e_2} + c * R_{12}^{e_3}$$

where the coefficients a , b , c , and the stress-ratio exponents are selected by the user.

In using the FSD method, it is important to understand the "closeness" of the resulting design to that of the minimum-weight design, as well as the convergence characteristics of the design solution. Frequently, for example, the method produces consecutive designs with very minor overall weight changes but with significant changes in the corresponding individual member properties. It is not possible to give general rules regarding the convergence behavior. Successful application of the method, therefore, depends highly on engineering judgement.

5.10.2 Thermal Fully Stressed Design (TFSD)

The thermal-fully-stressed design method can be used to resize structures that are subjected to both mechanical and thermal loads. This method is intended for the case when thermal stresses are of the same order of magnitude as the mechanical stresses. It is assumed that the mechanical loads and thermal stresses remain constant during resizing. As demonstrated in reference 5-15, use of the TFSD method results in a faster convergence of the design solution than that produced by the FSD method in the presence of significant thermal stresses. This characteristic is because stresses due to thermal effects quite

often remain reasonably constant while the mechanical stresses change with changing design variables.

The TFSD method is based on the same resize algorithm that is described in section 5.10.1 for the FSD method. The TFSD method, however, only considers strength requirements based on Hill's criterion. For the one-dimensional case, the property margins of safety are calculated by

$$MSC(\text{Strength}) = [(F-f_{th})/f_m] - 1$$

where "f_m" is the applied mechanical stress, "f_{th}" is the applied thermal stress, and "F" is the allowable stress as defined by the MATERIAL data set (sec. 140 in the User's Manual).

Other similarities between the FSD and the TFSD methods are the use of specified design constraints and margin-of-safety design envelopes, as well as the design solution convergence characteristics.

The TFSD method can be used for resizing the following types of stiffness finite elements.

ROD SPAR PLATE COVER SROD SPLATE

To use the TFSD method, it is necessary that a "THERMAL" DESIGN data subset is input to define one or more thermal design loadcases.

5.10.3 Composite Optimization

The composite optimization method can be used to resize composite structure that is idealized by the ATLAS CPLATE and CCOVER stiffness finite elements. The structure to be optimized (strength resized) is considered to be divided into one or more design regions, each of which defines an optimization problem that is processed independently. The resizing process consequently involves the repeated solution of local weight optimization problems, each of which is related to a subset of the elements contained in a particular design region. As illustrated in figure 5-21, this subset of elements is identified as an optimization subregion.

A design region and its optimization subregion are defined by a SUBSET DEFINITION data set (sec. 156 in the User's Manual). The design region or the optimization region can contain one or more elements of the same type, each with the same number of laminas and identical fiber directions.

Resizing of all the composite elements within a design region is based on satisfying the user-specified strength and weight design criteria for those elements in the optimization subregion. By including only the "strength-critical" elements in the subregion, the number of calculations associated with an optimization problem can be reduced significantly.

Each optimization problem is solved by the DESIGN processor according to the following steps:

- a) The stresses in the "first" lamina of each subregion element are compared for each design loadcase to identify the most "strength critical" first lamina. Similarly, the most critical second, third, etc. laminas are identified. Either Hill's strength criterion or the Maximum Strain criterion can be used, each of which is based on the "stress calculated" data, and the allowable stresses defined by the MATERIAL data set. This criticality screening establishes the critical element and loadcase for each lamina as shown in figure 5-22. The resulting set of "most critical" laminas defines the strength constraints that must be satisfied during the solution phase.
- b) The solution phase uses a local optimization "math-programming" technique based on Zoutendijk's method to determine a minimum weight design for the "strength critical" laminate (ref. 5-16). Structural weight is used as the objective function. The lamina thicknesses (number of layers in each lamina) are the design variables subject to the selected strength criterion. In the method of feasible direction, linear programming is used to establish new thicknesses such that the increase in structural weight is minimized without violating the strength constraints. During resizing, it is assumed that the lamina fiber directions remain fixed and that the total laminate loads remain constant. All lamina thicknesses of the elements in the design region are modified as necessary. Only the number of layers associated with the laminas is allowed to vary. Redistribution of the internal loads is established by the "next" stress analysis.
- c) After completion of the "math-programming" phase, a new criticality screening (step a) is performed. If a new set of strength constraints is established, the solution and screening phases are repeated ten times or until the new and old set of critical elements and loadcases are in agreement. Convergence of the solution is generally considered to have been attained when the relative and

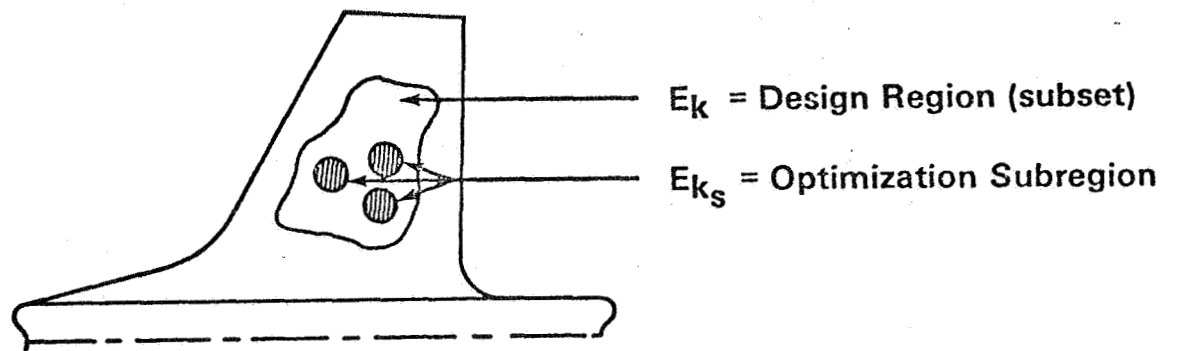


Figure 5-21. Regional Composite Optimization

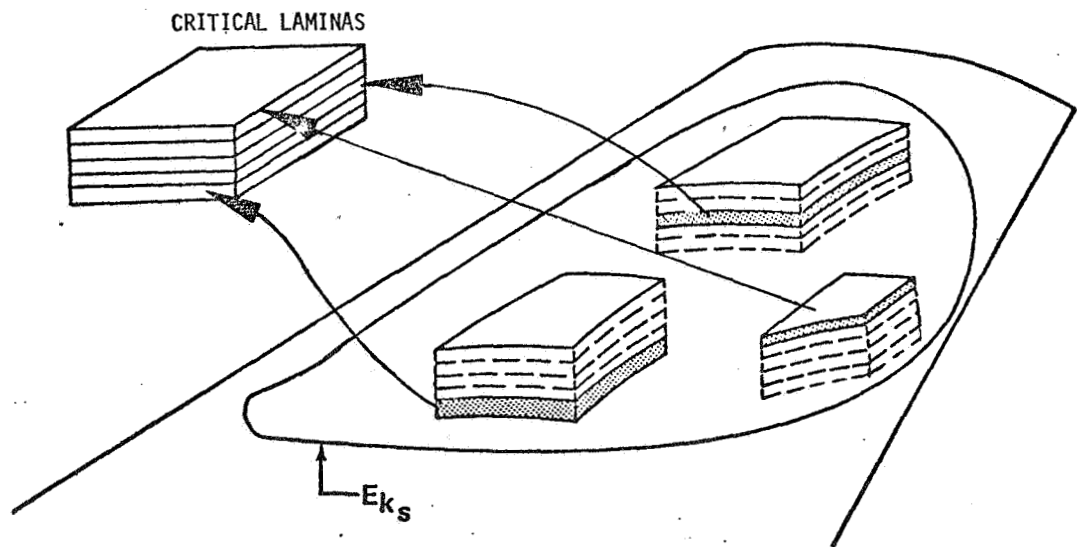


Figure 5-22. Strength Critical Laminate Screening

absolute change in the structural weight is less than .001 in three consecutive iterations.

The overall logic used in solving the composite optimization problem is illustrated in figure 5-23.

In addition to the general types of design constraints that can be specified by the DESIGN data set, as described previously, it is also possible to equate thicknesses for different laminas of composite structure. Use of this constraint capability allows "balanced" laminates to result from the resize calculations (e.g., equal lamina thicknesses in the $\pm 45^\circ$ directions).

5.10.4 Element Property Smoothing

The element-property smoothing capability allows the element properties to be modified without making changes directly to the input STIFFNESS data set. Use of this capability is intended primarily for "smoothing" the element properties resulting from an automated resizing in order to satisfy practical constraints, and to influence the design solution convergence characteristics.

Selected properties of selected elements can be made equal to one of the following.

- a) User-specified values provided in the "SMOOTHING" DESIGN data subset
- b) Those properties associated with a certain specified element
- c) The corresponding maximum property values associated with a specified subset (or region) of elements

It should be noted that previous calculation of element stresses is not required for use of this capability. Only the "SMOOTHING" DESIGN data subset is required in the model-definition data after a STIFFNESS data set is provided.

A capability that is closely related to the "smoothing" option is the "flutter execution" option of the DESIGN processor. In this case, structural properties of selected elements (or regions) are modified according to specified factors. This capability also represents a way of changing element properties without making changes directly to the input STIFFNESS data set (User's Manual; sec. 212).

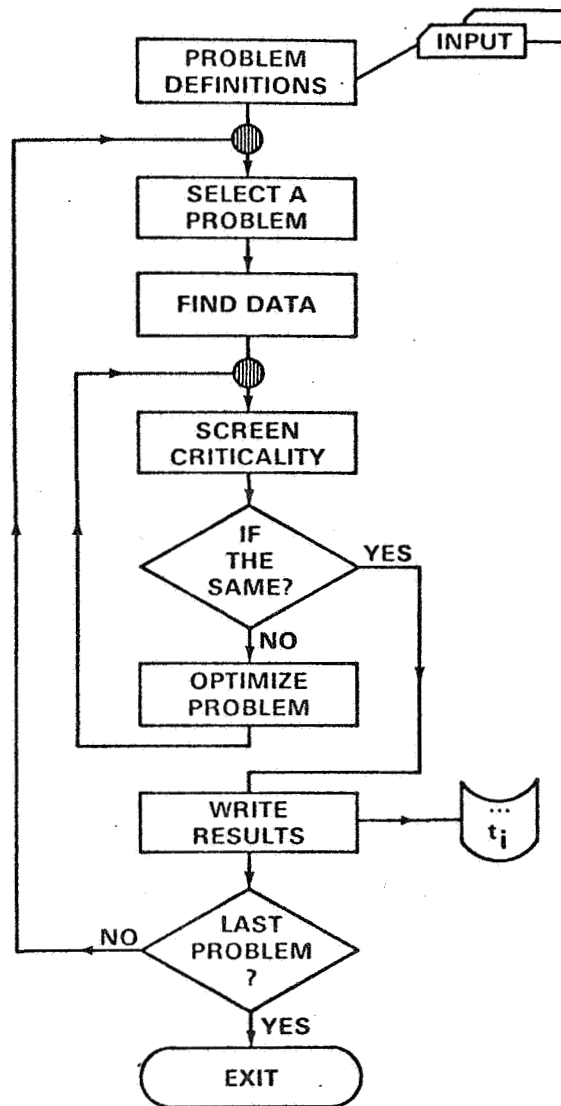


Figure 5 -23. Composite Optimization Problem Solution Logic

6.0 EXECUTION CONTROL DATA

The second phase of the problem-solving process requires the definition of ATLAS execution-control data to specify how the problem defined by the model-definition data (sec. 5) is to be solved. Basically, the ATLAS control data specify the method of analysis, the sequence of computational steps, and the printed/plotted displays of selected data to be generated.

All ATLAS System execution directives used to create a control deck are written in a user-oriented executive language. Optionally, FORTRAN and SNARK language statements may be intermixed with ATLAS directives to create a control deck for performing special analyses or for interfacing ATLAS data with other computer programs. These functions are described further in section 9. Generally, most problems can be solved by using only ATLAS directives (statements) in the execution-control data.

Having defined a Control deck, it may be used to execute ATLAS in a batch, on-line or mixed computing environment. Alternatively, the definition and execution of ATLAS control statements can be performed interactively. At execution time, an ATLAS Control Program is created automatically from the user-supplied execution-control data.

Functions that can be performed interactively during execution of ATLAS include

- a) Define, interrogate, and/or modify data files and execution-control files
- b) Execute selected ATLAS System modules to perform computations and to manage problem data
- c) Create and manipulate plots of selected data on a graphics terminal screen

Descriptions of the various execution-control options are presented in the following sections.

<u>Section</u>	<u>Description</u>
6.1	Control Program Functions
6.2	Printed Output
6.3	Graphical Output
6.4	Interactive Processing

Detailed descriptions of the ATLAS execution-control directives are presented in section 200 of the User's Manual.

6.1 CONTROL PROGRAM FUNCTIONS (User's Manual; sec. 200)

The ATLAS control statements used to create an execution-control deck follow the general conventions of FORTRAN (e.g., each statement must be written within columns 7 through 72 with optional line continuations). Each control statement has the following general format: Functional Descriptor (Plist)

The "Functional Descriptor" is one or more key-words that identify one or several ATLAS modules to be executed when the directive is processed. "Plist" is a list of parameters that is passed to the modules as they are being executed. These parameters are used at execution time for three purposes:

- a) Select execution options of a module
- b) Change default values of parameters as initially set by the module
- c) Specify numeric or alphanumeric information to be used during execution

The ATLAS execution-control statements, as summarized in table 6-1, are described in detail in section 200 of the User's Manual.

Each ATLAS control statement initiates one or more execution steps in the solution of the problem defined by the model-definition data deck. Functions performed by the control statements are as follows:

- a) Establish Executive Control -- Identify the ATLAS execution Control Program (control deck) and the mode of execution. The first and last statements of a control deck must be the "BEGIN CONTROL" and "END CONTROL" directives, respectively. A single-word "name" can be assigned to the Control Program, and optionally, a "text string" can be assigned by use of the PROBLEM ID statement. The INTERACTIVE CONTROL statement is used to initiate interactive ATLAS processing by use of a remote computer terminal. After execution control is transferred to the terminal, execution-directives are entered by the keyboard for on-line problem solving as described in section 6.4. If the INTERACTIVE CONTROL statement is not used, execution of the ATLAS job is performed in either the batch mode or the on-line mode.
- b) Preprocess Data -- Read, decode, and interrogate the model-definition data and generate problem data as requested. These functions are performed by the ATLAS

Table 6-1. Summary of the ATLAS Execution-Control Statements

BEGIN CONTROL <MATRIX> PROGRAM <name>
<n> CALL FILEADD (fel, {Filename})
<n> CALL PRNTCAT
<n> CALL REQFL (coresize)
CHANGE ID (text)
<n> END <CONTROL PROGRAM>
ERROR PROCEDURE
<n> EXECUTE {Processor Postprocessor} <(Plist)>
<n> INDEX FILES <(Filename)>
<n> INTERACTIVE CONTROL
<n> LOAD FILES <(<Savefile=REWIND,> Savefile,<Filename><.OPTION={1 2}>>>
<n> LOAD MATRIX (<Savefile=REWIND,> Savefile, Filename, Mlist<.OPTION={1 2}>>
<n> PERFORM {Procedure} <(Plist)>
<n> PRINT {MATRIX MATRIXID} (Filename, Mlist)
<n> PRINT {INPUT OUTPUT} ({Postprocessor}<, Plist>)
PROBLEM ID (text)
<n> PURGE FILES <(Filename)>
<n> PURGE MATRIX (FILE=Filename, Mlist)
<n> PURGE MATRIX <(Mlist)>
<n> READ INPUT <(<I=File,><MODE1 MODE2>>>
<n> RENAME MATRIX (Old1=new1, Old2=new2, . . .)
<n> SAVE FILES <(<Savefile=REWIND,> Savefile,<Filename>>>
<n> SAVE MATRIX (<Savefile=REWIND,> Savefile, Filename, Mlist)
USER COMMON (list)

Notation:

- {} indicates that one of the enclosed parameters must be input
- <> identifies a parameter that has a default value
- Filename -- The 7-character name of a random-access file associated with a particular ATLAS module (e.g., STIFRNF for the STIFFNESS processor)
- Processor, Postprocessor -- The name of an ATLAS module (e.g., the STRESS processor)
- PARAM -- All upper-case denotes a key-word
- Param -- Only the first character in upper case denotes either a key-word or a previously defined parameter
- param -- All lower case is user defined
- Plist -- A sequence of parameters denoting user-selected options
- Mlist -- List of matrix names associated with a Filename
- Savefile -- The name of an ATLAS file used for execution restart (e.g., SAVESF and SAVES1).

preprocessors, execution of which is initiated by the READ INPUT statement.

- c) Perform Computations -- Perform technical computations, as required for the problem solution, by executing one or more of the ATLAS processors. A processor is a named module that performs calculations associated with a particular engineering technology (e.g., LOADS, STRESS, VIBRATION, and FLUTTER). Execution of a processor is initiated by an "EXECUTE Processor" statement. Generally, a number of processors must be executed in sequence to perform an analysis. Certain standard types of analysis are performed most conveniently by use of the standard catalogs (sequences) of ATLAS statements. Execution of a control-procedure catalog is initiated by a "PERFORM Procedure" statement. Generation of a reduced stiffness matrix named KRED, for example, can be accomplished by the ATLAS statement

PERFORM K-REDUCE

which is equivalent to the following sequence of statements:

```
EXECUTE STIFFNESS (RUN=STIFFNESS,SET=1,LUMP=0.0)
EXECUTE MERGE (STIFFNESS,SET=1,STAGE=1,K11=11,K12=12,K22=22)
EXECUTE CHOLSKY (DEFO,K11,FK12,K12)
EXECUTE MULTIPLY (KRED=[ K22-FK12(T)*FK12])
```

- d) Postprocess Data -- Extract, format, and display (print/plot) the model-definition input data, analysis results, and data matrices. Selected data printouts are generated by the PRINT statements, whereas data plots are created by the "EXECUTE EXTRACT" and "EXECUTE GRAPHICS" statements. In general, there is a "PRINT INPUT" statement and a "PRINT OUTPUT" statement corresponding to each technical processor. All technical-data graphical displays, however, are created by using the common ATLAS graphics software package.
- e) Manage Data Base -- Establish a job-execution checkpoint or restart and manipulate data matrices established in the ATLAS data base. Selected information in the ATLAS data base can be saved for subsequent restart of job processing at an intermediate step. To restart job execution, the data are restored (loaded) into the ATLAS data base. These functions are performed by using the "SAVE" and "LOAD" statements, respectively. In each case, an external storage device (e.g., permanent file or magnetic tape) must be identified by the job control

cards described in section 7. The ERROR PROCEDURE statement is used to identify the beginning of a sequence of control statements that are to be executed only if a fatal error occurs during execution. Generally, the "error procedure" statements are used to generate select data printouts and to save the ATLAS data base for subsequent restart. Other control statements (e.g., INDEX, PURGE, and RENAME) are used primarily to manipulate the data matrices during large problem solving.

The following execution-control deck for performing a stress analysis illustrates the use of some of the ATLAS statements. Other examples are described in section 8.

```
BEGIN CONTROL PROGRAM EXAMPLE
PROBLEM ID (TYPICAL FORMAT OF AN EXECUTION-CONTROL DECK)
READ INPUT
PRINT INPUT (NODAL)
PRINT INPUT (STIFFNESS)
PERFORM STRESS
PRINT INPUT (BC)
PRINT OUTPUT (STRESSES)
SAVE FILES
END CONTROL PROGRAM
```

In this example, the model-definition data are preprocessed by the READ INPUT statement prior to requests for formatted printout of the NODAL and STIFFNESS input data. Stress-analysis computations are processed via the "PERFORM" cataloged control procedure. Options were selected to generate boundary condition and stress-data printout, and to save all ATLAS data for subsequent restart of job execution. It should be noted that all postprocessing activities (e.g., data printout and data saving) are performed only if requested by the execution-control data.

Those processors which must be executed to perform standard types of analysis using the ATLAS System are identified in table 6-2. Processors which may be executed in support of typical analyses are also shown in table 6-2. Because of the many analysis options provided by ATLAS, the sequence of module executions is best illustrated by the sample execution-control decks described in section 8. In all cases, execution of the postprocessors to generate print/plot displays is optional.

Table 6-2. Processor/Postprocessor Executions for Standard Types of Analyses ①

Processors and Postprocessors	Module Executions for Standard Analyses			
	Stress	Weights	Vibration	Flutter
ADDINT				✓
AFI				*
BC +	*	*	*	*
BUCKLING	*			
CHOLSKY	✓		✓	✓
DESIGN	*			*
DUBLAT				*
EXTRACT +	*	*	*	*
FLEXAIR				*
FLUTTER				✓
FREEBODY +	*			
GRAPHICS +	*	*	*	*
INTERACT +	*	*	*	*
INTERPOLATION				✓
LOADS	✓			
MACHBOX				*
MASS		✓	✓	✓
MATERIAL +	*	*	*	*
MERGE	✓		✓	✓
MULTIPLY	✓		✓	✓
NODAL +	*	*	*	*
REACTION +	*			
RH03				*
STIFFNESS	✓	*	✓	✓
STRESS	✓			
VIBRATION			✓	*

- ①
- A cross (+) identifies a postprocessor module only
 - A check (✓) identifies a processor that must be executed for the corresponding type of analysis.
 - An asterisk (*) identifies a processor or postprocessor that may be used to perform the corresponding analysis type.
 - Execution of all postprocessors for print/plot displays is optional.
 - All modules shown in the table or any combination thereof may be executed during an ATLAS job. The corresponding input data sets (ref. table 5-1), however, must be provided as necessary.

6.2 PRINTED OUTPUT

Two general types of printout are generated during processing of an ATLAS job.

- a) Automatic Printout
- b) Requested Printout

Printed output that is generated automatically include tracking of the execution steps performed during the problem solution, as well as certain solution accuracy checks. Formatted printout of selected input data and analysis results are generated only as requested by the user via "PRINT" statements in the execution-control deck (see sec. 6.1).

6.2.1 Automatic Printout

The types of printout that are generated automatically include the following:

- a) Execution-Control Deck Echo and Compilation
- b) Control Statement Echo at Module Execution Time
- c) Data Deck Echo
- d) ERROR and WARNING Messages
- e) Solution Accuracy Checks

Descriptions of these printed outputs are presented below.

6.2.1.1 Execution-Control Deck Echo and Compilation

The first activity performed automatically during execution of an ATLAS job is compilation of the execution-control deck. This activity is necessary to convert the control statements, as input in a user oriented language, into computer-sensible execution directives. Three blocks of printout are generated, as typified in figure 6-1 for a vibration analysis.

- a) The execution-control deck is printed as it was input (fig. 6-1a)
- b) Any execution control procedures referenced by the control deck are interpreted. That is, each PERFORM statement is replaced automatically by its corresponding sequence of explicit ATLAS statements (fig. 6-1b)
- c) The ATLAS-language statements are precompiled (translated) into FORTRAN-equivalent statements to form an ATLAS Control Program, which is then compiled and loaded for immediate execution (fig. 6-1c).

```

*****
*
*   ATLAS CATALOG INTERPRETER
*
*****
CARD
1      BEGIN CONTROL PROGRAM SAMPLE3
2      PROBLEM ID(SAMPLE CONTROL PROGRAM COMPILATION)
3      READ INPUT
4      EXECUTE MASS(OPTION=2)
5      PERFORM K-REDUCE
6      EXECUTE VIBRATION(MASS=MDC001A,STIF=KRED,NFREQS=5,NMODES=5)
7      PRINT OUTPUT(VIBRATION)
8      END CONTROL PROGRAM

```

(a) Echo of Execution-Control Deck

BCS-MACRO PROCESSOR FOR THE CDC 6600 (FTN VERSION)
 RUN DATE 77/10/31. TIME 20.17.23

```

*****
*
*   ATLAS PRECOMPILER (FTN VERSION 4.0)
*
*****

```

```

CARD      1...5  6 7                                73....80.....
1          BEGIN CONTROL PROGRAM SAMPLE3
2          PROBLEM ID(SAMPLE CONTROL PROGRAM COMPILATION)
3          READ INPUT
4          EXECUTE MASS(OPTION=2)
5          C
6          C      PERFORM K-REDUCE
7          C
8          C*****
9          C PURPOSE      GENERATE A REDUCED STIFFNESS MATRIX
10         C
11         C METHOD        1. GENERATE ELEMENT STIFFNESS MATRICES
12         C                2. ASSEMBLE THE STRUCTURAL STIFFNESS MATRIX
13         C                3. CALCULATE THE REDUCED STIFFNESS MATRIX
14         C
15         C RESULT       1. KRED ----- THE REDUCED STIFFNESS MATRIX
16         C
17         C SCRATCH MATRICES
18         C      FK12,K11,K12,K22
19         C*****
20         EXECUTE STIFFNESS(PUN=STIFFNESS,SET=1,LUMP=0.0)
21         EXECUTE MERGE(STIFFNESS,SET=1,STAGE=1,K11=11,K12=12,K22=22)
22         EXECUTE CHOLESKY(DEFD,K11,FK12,K12)
23         EXECUTE MULTIPLY(KRED=K22-FK12IT)*FK12)
24         EXECUTE VIBRATION(MASS=MDC001A,STIF=KRED,NFREQS=5,NMODES=5)
25         PRINT OUTPUT(VIBRATION)
26         END CONTROL PROGRAM

```

END PRECOMPIATION AT 20.17.24 (.267 SECONDS CP TIME USED.)

(b) Precompiled Control Program

```

PROGRAM SAMPLE3  74/74  OPT=1 ROUND=->/          FTN 4.6  428      77/10/31. 20.17.27      PAGE    1
1          OVERLAY (CONTROL,1,0)
2          PROGRAM SAMPLE3
3          COMMON /KERROR/ KERROR,KWARN,IPTU
4          COMMON /ONPARS/ KONPAR,CONPARS(30,2),KLABEL,LABEL(30)
5          COMMON /KOBUFF/ KOBUFF(2),SAVESS,SAVESS1,SAVESS2,SAVESS3,SAVESS4,
6          XSC0SSSF,SC01SSF,SC02SSSF,SC03SSSF,SC04SSSF,DBASFIL
7          INTEGER SAVESS,SAVESS1,SAVESS2,SAVESS3,SAVESS4,SC00SSSF,SC01SSF,SC
8          X02SSSF,SC03SSSF,SC04SSSF,DBASFIL
9          COMMON /KORDN/ KORDN,KORDN1,KORDN2,KORDN3,KORDN4,KORDN5,KORDN6,KORDN7,KORDN8,KORDN9,KORDN10,
10         X,DATARNF,DESIRNF,DUHLNF,INTERNF,LOADNF,MACHNF,MASSNF,MERGRNF,M
11         X
12         X
13         X
14         X
15         X
16         X
17         X
18         X
19         X
20         X
21         X
22         X
23         X
24         X
25         X
26         X
27         X
28         X
29         X
30         X
31         X
32         X
33         X
34         X
35         X
36         X
37         X
38         X
39         X
40         X
41         X
42         X
43         X
44         X
45         X
46         X
47         X
48         X
49         X
50         X
51         X
52         X
53         X
54         X
55         X
56         X
57         X
58         X
59         X
60         X
61         X
62         X
63         X
64         X
65         X
66         X
67         X
68         X
69         X
70         X
71         X
72         X
73         X
74         X
75         X
76         X
77         X
78         X
79         X
80         X
81         X
82         X
83         X
84         X
85         X
86         X
87         X
88         X
89         X
90         X
91         X
92         X
93         X
94         X
95         X
96         X
97         X
98         X
99         X
100        X
101        X
102        X
103        X
104        X
105        X
106        X
107        X
108        X
109        X
110        X
111        X
112        X
113        X
114        X
115        X
116        X
117        X
118        X
119        X
120        X
121        X
122        X
123        X
124        X
125        X
126        X
127        X
128        X
129        X
130        X
131        X
132        X
133        X
134        X
135        X
136        X
137        X
138        X
139        X
140        X
141        X
142        X
143        X
144        X
145        X
146        X
147        X
148        X
149        X
150        X
151        X
152        X
153        X
154        X
155        X
156        X
157        X
158        X
159        X
160        X
161        X
162        X
163        X
164        X
165        X
166        X
167        X
168        X
169        X
170        X
171        X
172        X
173        X
174        X
175        X
176        X
177        X
178        X
179        X
180        X
181        X
182        X
183        X
184        X
185        X
186        X
187        X
188        X
189        X
190        X
191        X
192        X
193        X
194        X
195        X
196        X
197        X
198        X
199        X
200        X
201        X
202        X
203        X
204        X
205        X
206        X
207        X
208        X
209        X
210        X
211        X
212        X
213        X
214        X
215        X
216        X
217        X
218        X
219        X
220        X
221        X
222        X
223        X
224        X
225        X
226        X
227        X
228        X
229        X
230        X
231        X
232        X
233        X
234        X
235        X
236        X
237        X
238        X
239        X
240        X
241        X
242        X
243        X
244        X
245        X
246        X
247        X
248        X
249        X
250        X
251        X
252        X
253        X
254        X
255        X
256        X
257        X
258        X
259        X
260        X
261        X
262        X
263        X
264        X
265        X
266        X
267        X
268        X
269        X
270        X
271        X
272        X
273        X
274        X
275        X
276        X
277        X
278        X
279        X
280        X
281        X
282        X
283        X
284        X
285        X
286        X
287        X
288        X
289        X
290        X
291        X
292        X
293        X
294        X
295        X
296        X
297        X
298        X
299        X
300        X
301        X
302        X
303        X
304        X
305        X
306        X
307        X
308        X
309        X
310        X
311        X
312        X
313        X
314        X
315        X
316        X
317        X
318        X
319        X
320        X
321        X
322        X
323        X
324        X
325        X
326        X
327        X
328        X
329        X
330        X
331        X
332        X
333        X
334        X
335        X
336        X
337        X
338        X
339        X
340        X
341        X
342        X
343        X
344        X
345        X
346        X
347        X
348        X
349        X
350        X
351        X
352        X
353        X
354        X
355        X
356        X
357        X
358        X
359        X
360        X
361        X
362        X
363        X
364        X
365        X
366        X
367        X
368        X
369        X
370        X
371        X
372        X
373        X
374        X
375        X
376        X
377        X
378        X
379        X
380        X
381        X
382        X
383        X
384        X
385        X
386        X
387        X
388        X
389        X
390        X
391        X
392        X
393        X
394        X
395        X
396        X
397        X
398        X
399        X
400        X
401        X
402        X
403        X
404        X
405        X
406        X
407        X
408        X
409        X
410        X
411        X
412        X
413        X
414        X
415        X
416        X
417        X
418        X
419        X
420        X
421        X
422        X
423        X
424        X
425        X
426        X
427        X
428        X
429        X
430        X
431        X
432        X
433        X
434        X
435        X
436        X
437        X
438        X
439        X
440        X
441        X
442        X
443        X
444        X
445        X
446        X
447        X
448        X
449        X
450        X
451        X
452        X
453        X
454        X
455        X
456        X
457        X
458        X
459        X
460        X
461        X
462        X
463        X
464        X
465        X
466        X
467        X
468        X
469        X
470        X
471        X
472        X
473        X
474        X
475        X
476        X
477        X
478        X
479        X
480        X
481        X
482        X
483        X
484        X
485        X
486        X
487        X
488        X
489        X
490        X
491        X
492        X
493        X
494        X
495        X
496        X
497        X
498        X
499        X
500        X
501        X
502        X
503        X
504        X
505        X
506        X
507        X
508        X
509        X
510        X
511        X
512        X
513        X
514        X
515        X
516        X
517        X
518        X
519        X
520        X
521        X
522        X
523        X
524        X
525        X
526        X
527        X
528        X
529        X
530        X
531        X
532        X
533        X
534        X
535        X
536        X
537        X
538        X
539        X
540        X
541        X
542        X
543        X
544        X
545        X
546        X
547        X
548        X
549        X
550        X
551        X
552        X
553        X
554        X
555        X
556        X
557        X
558        X
559        X
560        X
561        X
562        X
563        X
564        X
565        X
566        X
567        X
568        X
569        X
570        X
571        X
572        X
573        X
574        X
575        X
576        X
577        X
578        X
579        X
580        X
581        X
582        X
583        X
584        X
585        X
586        X
587        X
588        X
589        X
590        X
591        X
592        X
593        X
594        X
595        X
596        X
597        X
598        X
599        X
600        X
601        X
602        X
603        X
604        X
605        X
606        X
607        X
608        X
609        X
610        X
611        X
612        X
613        X
614        X
615        X
616        X
617        X
618        X
619        X
620        X
621        X
622        X
623        X
624        X
625        X
626        X
627        X
628        X
629        X
630        X
631        X
632        X
633        X
634        X
635        X
636        X
637        X
638        X
639        X
640        X
641        X
642        X
643        X
644        X
645        X
646        X
647        X
648        X
649        X
650        X
651        X
652        X
653        X
654        X
655        X
656        X
657        X
658        X
659        X
660        X
661        X
662        X
663        X
664        X
665        X
666        X
667        X
668        X
669        X
670        X
671        X
672        X
673        X
674        X
675        X
676        X
677        X
678        X
679        X
680        X
681        X
682        X
683        X
684        X
685        X
686        X
687        X
688        X
689        X
690        X
691        X
692        X
693        X
694        X
695        X
696        X
697        X
698        X
699        X
700        X
701        X
702        X
703        X
704        X
705        X
706        X
707        X
708        X
709        X
710        X
711        X
712        X
713        X
714        X
715        X
716        X
717        X
718        X
719        X
720        X
721        X
722        X
723        X
724        X
725        X
726        X
727        X
728        X
729        X
730        X
731        X
732        X
733        X
734        X
735        X
736        X
737        X
738        X
739        X
740        X
741        X
742        X
743        X
744        X
745        X
746        X
747        X
748        X
749        X
750        X
751        X
752        X
753        X
754        X
755        X
756        X
757        X
758        X
759        X
760        X
761        X
762        X
763        X
764        X
765        X
766        X
767        X
768        X
769        X
770        X
771        X
772        X
773        X
774        X
775        X
776        X
777        X
778        X
779        X
780        X
781        X
782        X
783        X
784        X
785        X
786        X
787        X
788        X
789        X
790        X
791        X
792        X
793        X
794        X
795        X
796        X
797        X
798        X
799        X
800        X
801        X
802        X
803        X
804        X
805        X
806        X
807        X
808        X
809        X
810        X
811        X
812        X
813        X
814        X
815        X
816        X
817        X
818        X
819        X
820        X
821        X
822        X
823        X
824        X
825        X
826        X
827        X
828        X
829        X
830        X
831        X
832        X
833        X
834        X
835        X
836        X
837        X
838        X
839        X
840        X
841        X
842        X
843        X
844        X
845        X
846        X
847        X
848        X
849        X
850        X
851        X
852        X
853        X
854        X
855        X
856        X
857        X
858        X
859        X
860        X
861        X
862        X
863        X
864        X
865        X
866        X
867        X
868        X
869        X
870        X
871        X
872        X
873        X
874        X
875        X
876        X
877        X
878        X
879        X
880        X
881        X
882        X
883        X
884        X
885        X
886        X
887        X
888        X
889        X
890        X
891        X
892        X
893        X
894        X
895        X
896        X
897        X
898        X
899        X
900        X
901        X
902        X
903        X
904        X
905        X
906        X
907        X
908        X
909        X
910        X
911        X
912        X
913        X
914        X
915        X
916        X
917        X
918        X
919        X
920        X
921        X
922        X
923        X
924        X
925        X
926        X
927        X
928        X
929        X
930        X
931        X
932        X
933        X
934        X
935        X
936        X
937        X
938        X
939        X
940        X
941        X
942        X
943        X
944        X
945        X
946        X
947        X
948        X
949        X
950        X
951        X
952        X
953        X
954        X
955        X
956        X
957        X
958        X
959        X
960        X
961        X
962        X
963        X
964        X
965        X
966        X
967        X
968        X
969        X
970        X
971        X
972        X
973        X
974        X
975        X
976        X
977        X
978        X
979        X
980        X
981        X
982        X
983        X
984        X
985        X
986        X
987        X
988        X
989        X
990        X
991        X
992        X
993        X
994        X
995        X
996        X
997        X
998        X
999        X
1000       X

```

CARD NO.	SEVERITY	DETAILS	DIAGNOSIS OF PROBLEM
38	I		RETURN STATEMENT APPEARS IN MAIN PROGRAM.
46	I		RETURN STATEMENT APPEARS IN MAIN PROGRAM.
59	I		ARGUMENT COUNT INCONSISTENT WITH PRIOR USAGE.
84	I		ARGUMENT COUNT INCONSISTENT WITH PRIOR USAGE.

(c) Compiled Control Program

Figure 6-1. Execution-Control Deck Echo and Compilation Printout

Diagnostic messages, if any, are printed either by the ATLAS precompiler or by the computer operating system used by the computer installation.

6.2.1.2 Control Statement Echo at Module Execution Time

Each ATLAS control statement is printed at execution time, as illustrated in figure 6-2. The following information is included:

- a) The PROBLEM ID as specified by the execution-control deck
- b) The version of the ATLAS System being executed
- c) The total central processing seconds utilized up to this step during the job processing
- d) Execution parameters, as specified by the user, to be used during execution of the particular ATLAS module
- e) Generally, the computer field length required for execution of the module is also displayed at this time.

6.2.1.3 Data Deck Echo

All input data records included in the model definition data deck are printed as they are read. Each data record is assigned an input record number and a card number as illustrated in the data deck echo shown in figure 6-3. Note that non-executable data records, such as data comments, are not assigned a consecutive input record number. Reading and printing the data deck is initiated by the READ INPUT execution-control statement.

6.2.1.4 ERROR and WARNING Messages

During problem execution, ATLAS performs an extensive number of data checks in an attempt to trap all possible anomalies. When the system detects an ambiguity in the data that can be resolved without user interaction, a warning message is printed and processing of the job continues without interruption. However, when a computer operating system error occurs, or when a fatal inconsistency is detected either in the data or in the user-selected execution logic, an error message is printed. In this case, only the execution directives included in an "ERROR PROCEDURE" within the execution-control deck are processed prior to terminating the job.

In all cases, the word WARNING or the word ERROR is included in the appropriate message issued to the output file. Each

```

SAMPLE OUTPUT LISTING
ATLAS 4.0 PROD   UD102
ELAPSED CP SECONDS   29.359
CALL OVERLAY 1,0 FROM FILE VIBROLF
PARAMETERS PASSED TO MODULE
*****
1      EXECUTE
2      VIBRATION
3      STIF      = KRED
4      MASS      = MDCOOLA
5      NFREQS    =          5
6      NMODES    =          5

FIELD LENGTH REQUIRED FOR THIS MODULE IS 115776 OCTAL

```

Figure 6-2. Example Printout of a Control Statement at Execution Time

```

*/ MODE2 /
BEGIN NODAL DATA
100 5. 10. 15.
101 6. 10. 18.
102 7. 15. 15.
103 8. 15. 18.
104 9. 20. 15.
105 10. 20. 18.
END NODAL DATA
END PROBLEM DATA

```

(a) Input Data Deck

INPUT RECORD	1	*/ MODE2 / CARD	1 (1)
INPUT RECORD	1	BEGIN NODAL DATA CARD	2 (1)
INPUT RECORD	2	100 5. 10. 15. CARD	3 (1)
INPUT RECORD	3	101 6. 10. 18. CARD	4 (1)
INPUT RECORD	4	102 7. 15. 15. CARD	5 (1)
INPUT RECORD	5	103 8. 15. 18. CARD	6 (1)
INPUT RECORD	6	104 9. 20. 15. CARD	7 (1)
INPUT RECORD	7	105 10. 20. 18. CARD	8 (1)
INPUT RECORD	8	END NODAL DATA CARD	9 (1)
INPUT RECORD	9	END PROBLEM DATA CARD	10 (1)

(b) Data Deck Echo

Figure 6-3. Model-Definition Data Deck Echo

message is printed as the situation is encountered and is intended to be self-explanatory. Example WARNING and ERROR messages are as follows:

WARNING -- COORDINATES OF NODE NUMBER 42 HAVE BEEN REDEFINED.

WARNING -- THE WARPING FACTOR IS .017; IT SHOULD NOT EXCEED 0.05
FOR BRICK FACES.

ERROR -- NODAL COORDINATES MUST BE INPUT AS DECIMAL NUMBERS.

ERROR -- ELEMENT LENGTHS MUST BE FINITE.

ERROR -- REENTRANT CORNERS ARE NOT ALLOWED FOR QUADRILATERAL
ELEMENTS.

ERROR -- LOADS HAVE BEEN SPECIFIED FOR INACTIVE NODAL FREEDOMS.

It should be noted that all anomalies encountered during postprocessing activities, (i.e., generation of requested printouts and plots) are identified as warning conditions so that further processing of the job is not terminated.

6.2.1.5 Solution Accuracy Checks

Many solution accuracy checks are performed automatically or are provided as execution options. In all cases, the results of the checks are identified in the printout.

The check performed automatically to identify ill-conditioning of the structural model is used to illustrate this type of printout.

When the MERGE processor is executed to form the gross structural stiffness matrix using the elemental stiffness matrices as its components, the correspondence between the rows of the merged free-free stiffness matrix partition (the 1,1 partition) and the nodal freedoms is printed automatically. Figure 6-4 illustrates this correspondence table.

When the structural equations of equilibrium are solved by the CHOLSKY processor, the free-free stiffness matrix partition is decomposed into the product of a lower-triangular matrix post-multiplied by its transpose. While performing this decomposition, the number of significant digits lost during the computations (the pivot decays) are calculated. As the degree of equation dependency increases, the number of significant digits lost during the solution also increases. When two or more of the equations are found to be dependent within the accuracy of the computer, the system of equations is singular. In this case, an

ERROR message is printed and job processing is terminated. The mathematical model must be modified to remove the equation dependencies so that a nontrivial solution can be performed.

The row numbers of the free-free stiffness matrix partition associated with all singularities and the maximum pivot decay, as detected by the CHOLESKY processor, are identified in the printout. Using the aforementioned row/freedom correspondence table printed by the MERGE processor, the nodal freedoms corresponding to the singularities and "local mechanisms" can be identified. For the example shown in figure 6-5, one significant digit was lost during the computations associated with row 60 of the stiffness matrix. According to figure 6-4, this row corresponds to the sixth freedom (the RZ freedom) of node number 11.

6.2.2 Requested Printout

Many options are provided for requesting formatted printout of selected model-definition input data, analysis results, and data matrices. In all cases, data printouts are generated only upon user request by including the appropriate "PRINT" control statements in the execution control data. In general, there is a "PRINT INPUT" statement associated with each input data set (each preprocessor), and a "PRINT OUTPUT" statement associated with each technical processor.

Examples of requested printout are presented in section 8 for a number of example problems.

6.3 GRAPHICAL OUTPUT (User's Manual; sec. 228)

The GRAPHICS postprocessor produces on-line graphical displays on the Tektronix model 4014 graphics console in addition to off-line plots on the Stromberg Carlson SC4020, the CALCOMP Model 763, and the GERBER drafting machine. All ATLAS technical data plots are created by using this module. Execution of the GRAPHICS postprocessor can be performed in the interactive mode and in the batch mode. The user is provided with a passive interactive graphics capability which enables plots to be viewed between problem-solution steps for verification of the technical analysis process.

```

*** PARTITION 1,1 ROW/FREEDOM CORRESPONDENCE ***
SET 1 STAGE 1
PROBLEM ID = SIMPLE 10 ELEMENT BEAM
DATE 78/04/11 PAGE 1
PARTITION 1,1 ROWS OF THE MERGED STIFFNESS MATRIX ARE ASSOCIATED WITH THE NODAL FREEDOMS AS SHOWN IN THIS TABLE.

```

PARTITION 1,1 ROW NUMBERS		NODAL FREEDOM/ROW CORRESPONDENCE						NODE NUMBER	
		NODAL FREEDOM NUMBER						USER	INTERNAL
***** - *****		1	2	3	4	5	6	*****	*****
1 =	4	1				2	3	4	1
5 =	10	5	6	7	8	9	10	2	2
11 =	16	11	12	13	14	15	16	3	3
17 =	22	17	18	19	20	21	22	4	4
23 =	28	23	24	25	26	27	28	5	5
29 =	34	29	30	31	32	33	34	6	6
35 =	40	35	36	37	38	39	40	7	7
41 =	46	41	42	43	44	45	46	8	8
47 =	52	47	48	49	50	51	52	9	9
53 =	58	53	54	55	56	57	58	10	10
59 =	60				59	60		11	11

RUN TIME FOR ABOVE OVERLAYX .352 CP SECONDS:

Figure 6-4. Partition 1,1 Row/Freedom Correspondence Table

```

SIMPLE 10 ELEMENT BEAM
ATLAS 4.0 PROD UD102
ELAPSED CP SECONDS 17.5981
CALL OVERLAY 1,0 FROM FILE CHOLDF
PARAMETERS PASSED TO MODULE
*****
1 EXECUTE
2 CHOLSKY
3 SOLVE
4 K11T
5 D11
6 TEMP
THE MAXIMUM PIVOT DECAY OCCURRED IN ROW 60 AND WAS 1.
RUN TIME FOR ABOVE OVERLAYX .228 CP SECONDS:

```

Figure 6-5. Pivot-Decay Solution Accuracy Message

The conversational dialog used during interactive-graphics processing is conducted via menus. These menus allow the user to select which plots are to be displayed on the scope of the console, generate different displays of selected data, and enlarge selected areas of on-line plots by zooming. Selected hardcopy plots can be generated on-line by using a hardcopy unit that is compatible with the Tektronix 4014. Additionally, selected plots viewed on the screen can be directed to one of the off-line plot devices (SC4020, CALCOMP or GERBER) to obtain a higher-quality hardcopy print. These on-line graphics facilities complement those capabilities provided by the GRAPHICS postprocessor for creation of plots.

The following two execution-control statements are required to generate ATLAS plots.

```
EXECUTE EXTRACT (Plist)
EXECUTE GRAPHICS (Plist)
```

The EXTRACT statement is used to select the input data and/or data previously calculated by another ATLAS processor that are to be displayed graphically. Parameters specified in the GRAPHICS statement "Plist" are used to identify how the data are to be displayed. Only one off-line plotting device can be identified per job and, when used, it must be identified by the PLOTFIL job control card as described in section 7. This single-device limitation is inherent to the computer operating system.

Those blocks of data that are required for generating particular plot types, and which must be extracted to create a plot, are identified by standard label subsets. A label subset name is specified in the EXTRACT statement "Plist" to identify which data components associated with the mathematical model are to be extracted. Table 6-3 identifies which label subset name should be used to create a particular type of plot. Each extracted data block is identified by a user-assigned label using the EXNAME parameter in the EXTRACT "Plist." Multiple plots can be generated from the block of data associated with an EXNAME.

The extracted data may be identified explicitly or they may be associated with user-selected geometric regions of the mathematical model. Regions are defined as node and/or element subsets of the model by input of a SUBSET DEFINITION data set as described in section 156 of the User's Manual. Some of the data components are dependent on loadcases, mass-distribution conditions, flutter conditions, etc. Extraction of data values associated with subsets of cases or conditions is controlled by the EXTRACT statement parameters as described in section 218 of

Table 6-3. Standard Label Subsets as Used to Create Data Plots

STANDARD LABEL-SUBSET NAME	DATA PLOT TO BE GENERATED (ref. Table 6-4)
BMODE	Buckling mode-shape displacements at retained nodes
DISGRID	Deformed structural grid or displacements on structural grid
DISNODE	Displacements at nodal locations without structural grid
KGRID	Structural finite-element grid
KPROP	Stiffness element properties on structural grid
LOADAB	Fuel/payload loadability diagrams
MGRID	Mass finite-element grid
NODES	Nodes without grid
SMS	"Strength designed" element- property margins of safety on structural grid
STRESS	Stresses on structural grid
TMS	"Thermal-designed" element- property margins of safety on structural grid
VGXF	Velocity-damping (V-g) and velocity-frequency (V-f) graphs
VMODE	Vibration mode-shape displacements at retained nodes

the User's Manual. By appropriate selection of label subsets, regions of the mathematical model (node/element subsets), and case/condition subsets, the user may extract any number of input and calculated data values for creating plots. An extracted data block may include anywhere from one data value up to all data values associated with a mathematical model.

A summary of the types of plots that can be created from the extracted technical data identified by the standard label subset names is shown in table 6-4. Descriptions and examples of the various display options are presented in section 6.3.1.

Execution parameters specified by the GRAPHICS statement "Plist" include the following:

- a) The EXNAME data block name to be used to generate the plot
- b) Plot unit; either on-line or an off-line plot device
- c) Plot type; either orthographic, contour, graph or matrix
- d) Plot size or scale
- e) Viewing position
- f) Plot labeling information

Detailed descriptions of the use and capabilities of the GRAPHICS postprocessor are presented in section 228 of the User's Manual.

6.3.1 Display Options

The following plot types, as identified in table 6-4, are described below by using example plots.

- a) Undeformed Geometry
- b) Deformed Geometry
- c) Scalar-Grid
- d) Contour
- e) Graph
- f) Matrix

6.3.1.1 Undeformed Geometry Plots

The geometry of the nodes and finite elements used to define structural and mass models can be plotted to any scale and relative to any viewing position as selected by the user. Pictorial, GLOBAL-axis, and exploded views of the model are created using orthographic projection.

Table 6-4. Technical-Data Plot Types

TECHNICAL DATA DISPLAY	PLOT TYPE						APPLICABLE STANDARD LABEL-SUBSET (LSUB) NAME(S)
	UNDEFORMED GEOMETRY	DEFORMED GEOMETRY	SCALAR-GRID	CONTOUR	GRAPH	MATRIX	
STRUCTURAL AND/OR MASS MODEL							
• Nodes	•						NODES, KPROP, KGRID, MGRID, DISNODE, DISGRID or STRESS
• Stiffness finite-element grid	•						KGRID, KPROP or STRESS
• Stiffness element properties			•	•			KPROP
• Mass finite-element grid	•						MGRID
• Exploded node/grid subsets	•						NODES, KPROP, KGRID, MGRID, DISNODE, DISGRID or STRESS
DISPLACEMENTS/STRESSES							
• Nodal displacements		•					DISNODE or DISGRID
• Deformed structural grid		•					DISGRID
• Element stresses			•	•	•		STRESS
STRUCTURAL RESIZE RESULTS							
• Stiffness element properties			•	•			KPROP
• Strength-designed margins of safety			•	•	•		SMS
• Thermal-designed margins of safety			•	•	•		TMS
MODE SHAPES							
• Vibration		•					VMODE
• Buckling		•					BMODE
FUEL/PAYLOAD MANAGEMENT							
• Loadability diagrams					•		LOADAB
FLUTTER SOLUTION							
• V-g and V-f graphs					•		VGVF
MATRIX DATA							
• Any ATLAS matrix						•	specified matrix name

Nodes can be displayed either as

- a) a network of points, or
- b) a network of points annotated with node numbers as shown in figure 6-6.

Stiffness and mass finite element grids can be displayed using any of the following options.

- a) Straight line segments denoting the nodal connectivity
- b) Same as (a) with node numbers
- c) Same as (a) with element numbers as shown in figure 6-7
- d) Same as (a) with element-type numbers (1=ROD, 2=BEAM, etc.)
- e) Same as (c) plus node numbers
- f) Same as (d) plus node numbers

Elements with offsets can be displayed either in the offset position or by connectivity of only the structural nodes used to define the element.

An "exploded view" of the nodes and/or stiffness and mass elements of a model can be created. The relative position (separation) and orientation of each component of the "exploded" model are specified for a number of different node and element subsets. An example is shown in figure 6-8. The plot options selected to display the individual node and/or element subsets can be different.

6.3.1.2 Deformed Geometry Plots

The deformed geometry of a statically loaded model, vibration mode shapes and general-instability buckling mode shapes can be plotted to any scale and relative to any viewing position as selected by the user. Pictorial and GLOBAL-axis views of the model are created using orthographic projection.

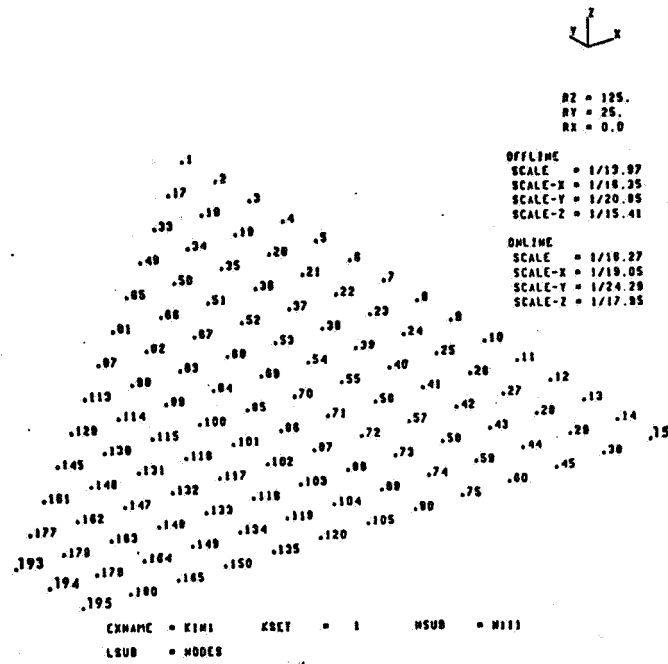


Figure 6-6. Point Plot of Nodes

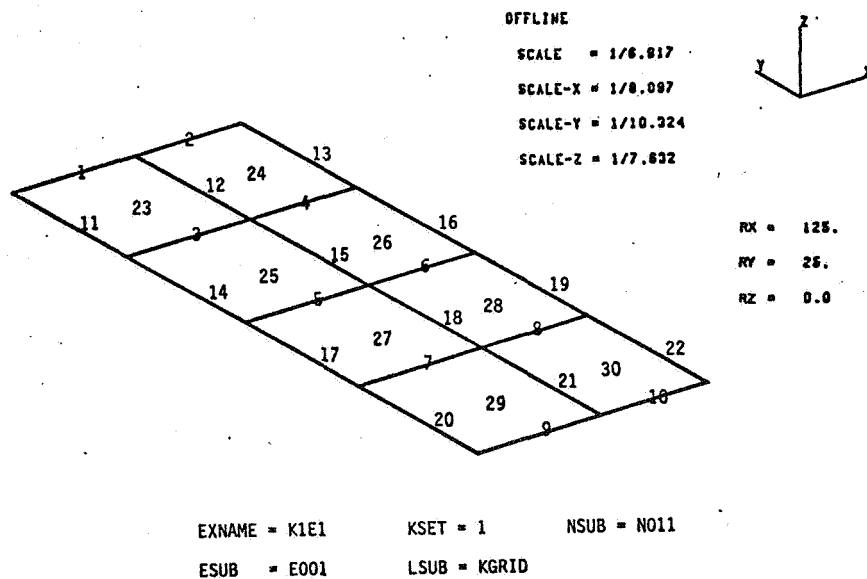


Figure 6-7. Element Grid with Element Numbers

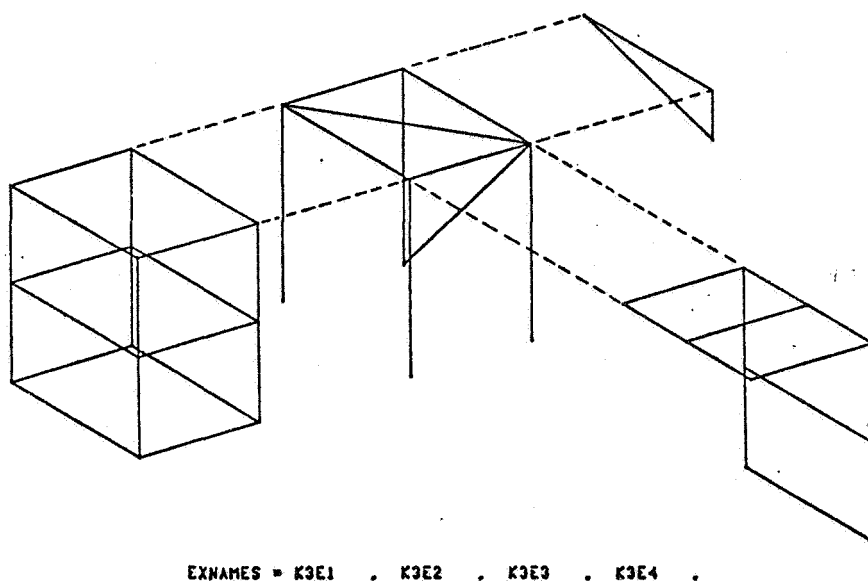


Figure 6-8. Exploded Geometry Plot

The options provided for creating deformed geometry plots are:

- a) Points plus vectors -- The points identify the undisplaced nodes, and the scaled vectors, which start at the nodes, represent the corresponding displacements (see fig. 6-9a).
- b) Undeformed grid plus vectors -- Straight line segments denoting the nodal connectivity of the undeformed model plus vectors that represent the nodal displacements (see fig. 6-9b).
- c) Deformed grid -- Dashed, straight line segments denoting the nodal connectivity of the deformed model as shown by figure 6-9c.
- d) Undeformed grid plus the deformed grid (fig. 6-9d).
- e) Undeformed and deformed grids plus vectors, as shown by figure 6-9e.

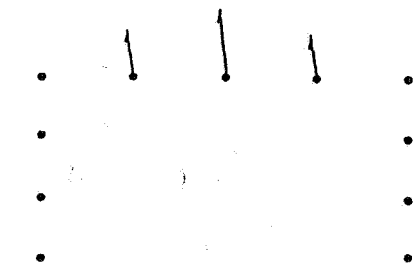
Additional options are provided for labeling the nodes and/or elements in the plots.

6.3.1.3 Scalar-Grid Plots

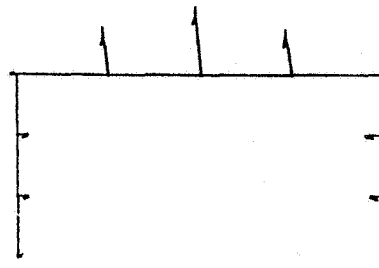
The following data values can be displayed as scalar ordinates superimposed on an orthographic plot of a structural element grid.

- a) Stiffness element properties
- b) Element stress components
- c) Element property margins of safety as calculated by the DESIGN processor

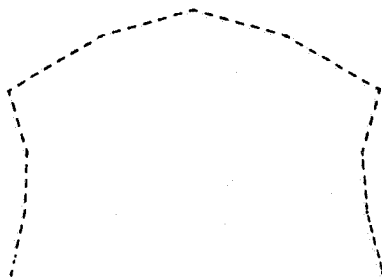
For a selected element type, either one property, one stress component for a selected loadcase, or one minimum margin of safety for a selected design cycle can be displayed as the scalar ordinate. The ordinates are plotted at right angles to the



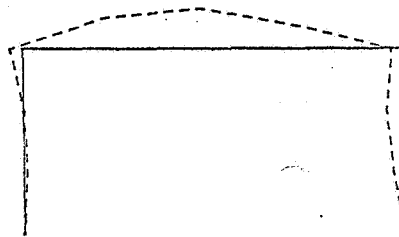
(a) Points plus Vectors



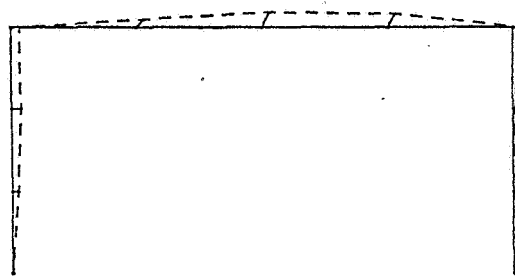
(b) Undeformed Grid
plus Vectors



(c) Deformed Grid



(d) Undeformed and
Deformed Grids



(e) Undeformed and Deformed Grids plus Vectors

Figure 6-9. Deformed Geometry Plot Options

display plane and at the element c.g. locations. The total display is reoriented automatically to a predefined position to generate a meaningful view of the data. The viewing position, plot scales, as well as node and/or element labeling are selected by the user. An example scalar-grid plot is shown by figure 6-10.

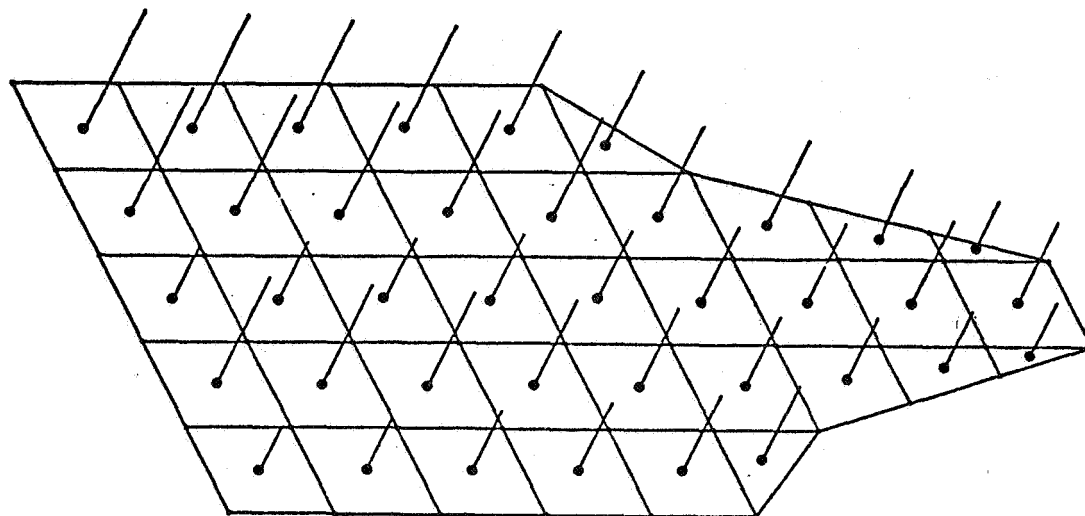


Figure 6-10. Example Scalar - Grid Plot

6.3.1.4 Contour Plots

The following data values can be displayed as isocurves superimposed on an orthographic plot of the structural element grid.

- a) Stiffness element properties
- b) Element stress components
- c) Element property margins of safety as calculated by the DESIGN processor

For a selected element type, either one property, one stress component for a selected loadcase, or one minimum margin of safety for a selected design cycle can be displayed. Each plotted isocurve depicts a constant level of the selected data value. Values of the scalar data component to be plotted are associated with the element c.g. locations. The viewing

position, plot scales, as well as the isocurve intervals are selected by the user.

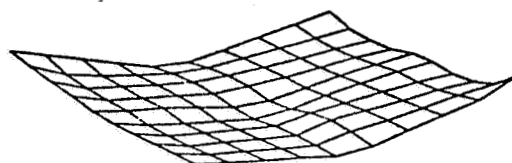
Two algorithms are provided for generating contour plots as described below.

- a) Rectangular Mesh -- The rectangular boundary of the contour plot is defined by the minimum and maximum coordinates of the data points. Data values are interpolated to the intersection points of a uniform grid within the rectangular boundary prior to generating isocurves. The uniform grid, displaced according to the interpolated data values, can be displayed in addition to the contour plot as shown in figure 6-11a.
- b) Triangular Mesh -- The boundary of the contour plot is defined by a sequence of nodes which, when connected in the specified order, establishes the boundary. Using the boundary nodes and the projected data points, the program automatically generates a mesh of adjacent triangles over the enclosed region. Data values are interpolated linearly during generation of the isocurves. The triangulated grid can be displayed in addition to the contour plot as shown in figure 6-11b.

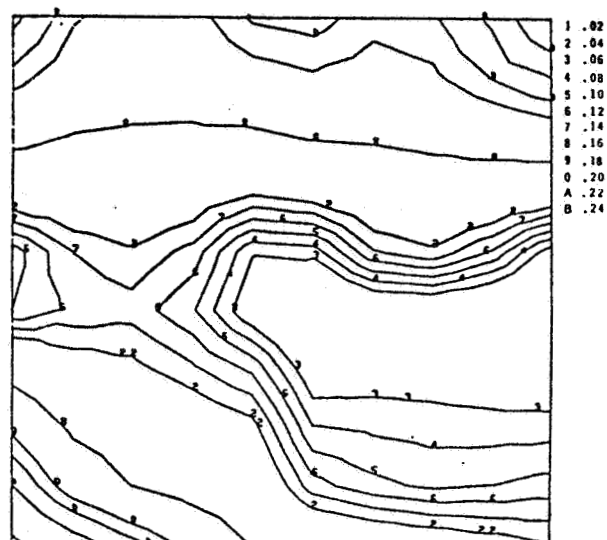
6.3.1.5 Graph Plots

Graphs of the following data values can be created.

- a) One stress component for a selected element type versus loadcase (see fig. 6-12a).
- b) One minimum margin of safety, as calculated by the DESIGN processor, for a selected element type versus design cycle
- c) Velocity-damping (V-g) and velocity-frequency (V-f) plots of flutter solutions for selected flight conditions as shown by figures 6-12b and 6-12c.
- d) Loadability diagrams showing how the c.g. location of the model moves with respect to user-specified passenger, cargo, and fuel loading management data (see fig. 6-12d).

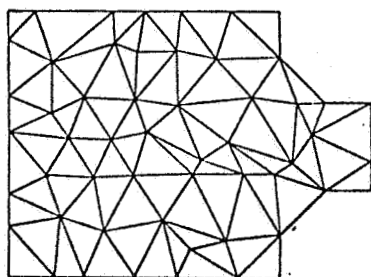


Displaced Grid

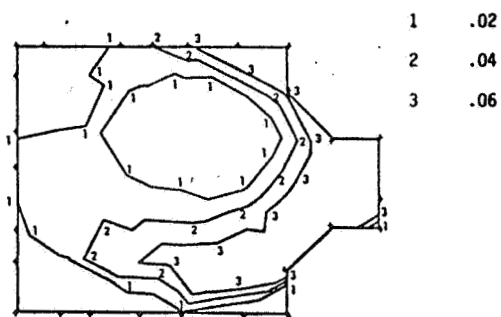


Contour Plot

(a) Rectangular Mesh and Boundary



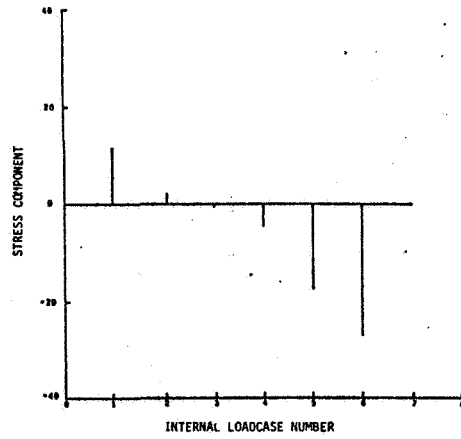
Triangulated Grid



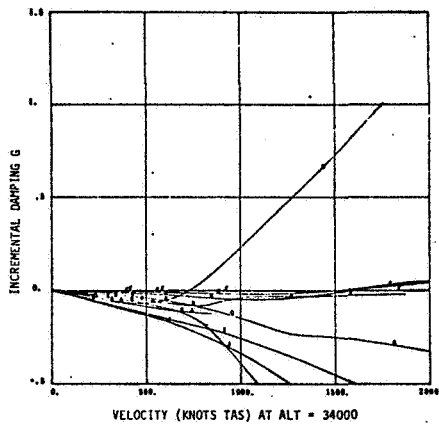
Contour Plot

(b) Triangular Mesh and Irregular Boundary

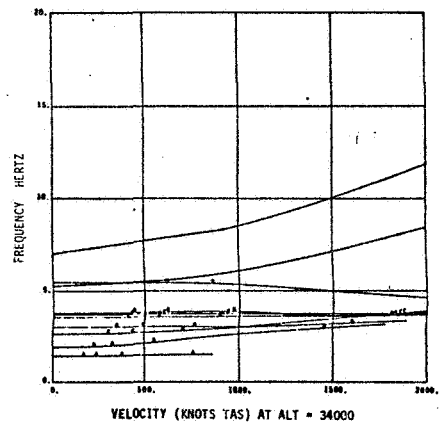
Figure 6-11. Contour Plot Options



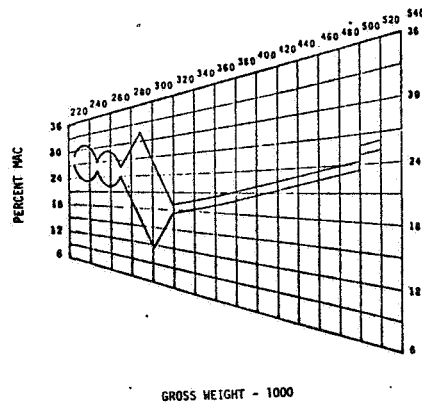
(a) Element Stress vs. Loadcase



(b) Flutter V-g Graph



(c) Flutter V-f Graph



(d) Loadability Diagram

Figure 6-12. Example Graph Plots

6.3.1.6 Matrix Data Plots

Any ATLAS data matrix can be displayed graphically as illustrated by figure 6-13.

6.3.2 Interactive Graphics

The GRAPHICS postprocessor can be executed interactively by using a Tektronix model 4014 graphics terminal. The on-line graphics facilities provided by the interactive-execution mode complement those graphical-display capabilities provided by the EXECUTE GRAPHICS statement. Those additional functions that can be performed interactively include the following:

- a) Display selected plots on the terminal screen
- b) Generate instant hardcopy plots of selected displays as shown on the terminal screen. A hardcopy unit that is compatible with the Tektronix 4014 must be used.
- c) Create new displays of selected plots via zooming and rotation.
- d) Enlarge selected regions of on-line plots by zooming.
- e) Select plots to be generated by one of the off-line plotting devices (SC4020, CALCOMP or GERBER)

If ATLAS is being executed by a graphics terminal, the interactive conversational mode is activated automatically for each EXECUTE GRAPHICS statement in the execution-control data deck. At this point, additional plot directives can be input using the keyboard.

Any of the plot types described in section 6.3.1 can be displayed and manipulated using the interactive graphics capabilities. The engineer conducts an interactive dialog with the GRAPHICS postprocessor by using a function menu, two plot directories, and a plot transformation menu as described below.

- a) Function Menu -- A list of execution functions, as shown in figure 6-14, from which user-selections are made on-line.
- b) GNAME Directory (G-D) -- The G-D contains a list of the user-specified plot group names, GNAMEs, as defined by EXECUTE GRAPHICS statements. Figure 6-15 shows a typical G-D. After a GNAME is selected from the G-D, the corresponding Plot ID Directory is displayed.

ROWS 100		COLUMNS 10				
	1	2	3	4	5	
1	1.	101.	201.	301.	401.	
2	2.	102.	202.	302.	402.	
3	3.	103.	203.	303.	403.	
4	4.	104.	204.	304.	404.	
5	5.	105.	205.	305.	405.	
6	6.	106.	206.	306.	406.	
7	7.	107.	207.	307.	407.	
8	8.	108.	208.	308.	408.	
9	9.	109.	209.	309.	409.	
10	10.	110.	210.	310.	410.	
11	11.	111.	211.	311.	411.	
12	12.	112.	212.	312.	412.	
13	13.	113.	213.	313.	413.	
14	14.	114.	214.	314.	414.	
15	15.	115.	215.	315.	415.	
16	16.	116.	216.	316.	416.	
17	17.	117.	217.	317.	417.	
18	18.	118.	218.	318.	418.	
19	19.	119.	219.	319.	419.	
20	20.	120.	220.	320.	420.	

Figure 6-13. Example Matrix Data Plot

FUNCTION MENU	
KEY	DESCRIPTION
FB,1	EXECUTE CURRENT MENU SELECTION(S)
FB,2	DISPLAY FUNCTION MENU
FB,3	DISPLAY GNAME DIRECTORY (G-D)
FB,4	DISPLAY PLOT ID DIRECTORY (P-I-D) FOR THE GNAME SELECTED FROM G-D
FB,5	DISPLAY REMAINING PLOTS FOR THIS STATEMENT ONLINE ONLY (ENTER FB,5 TWICE)
FB,6	DISPLAY NEXT PLOT ONLINE ONLY
FB,7	WRITE PLOTS SELECTED BY A STRING OF LP,N COMMANDS ONTO V/F WITHOUT ONLINE DISPLAY
FB,8	DISPLAY PLOT TRANSFORMATION MENU (P-T-M) TO ROTATE AND/OR ZOOM
FB,9	INPUT TRANSFORMATION PARAMETERS WITHOUT P-T-M DISPLAY
FB,10	ZOOM CURRENT PLOT VIA CROSSHAIRS
FB,11	DISPLAY DEFORMED GRID OR TRIANGULATED REGION FOR A CONTOUR PLOT
FB,12	RETURN TO ATLAS CONTROL PROGRAM
FB,13	STOP ATLAS EXECUTION (ENTER FB,13 TWICE)

Figure 6-14. Graphics Function Menu Options

GNAME DIRECTORY			
1. GEOMETRY	2. LOADS	3. STRESS	4. BUCKLING
5. VIBRATION	6. FLUTTER		

Figure 6-15. Typical GNAME Directory

PLOT ID DIRECTORY FOR GNAME=GEOMETRY	
1. WING,GEOMETRY	2. BODY,GEOMETRY
3. VTAIL,GEOMETRY	4. HTAIL,GEOMETRY
5. NACELLE,GEOMETRY	6. MODEL5,GEOMETRY,LC=DIVE
7. ROUGH,GEOMETRY,LC=TAXI	8. COND3,MODE SHAPE NO. 5
9. MASS2,GEOMETRY	10. EXPLODED GEOMETRY

Figure 6-16. Typical Plot ID Directory

- c) Plot ID Directory (P-I-D) -- A P-I-D is generated automatically for each GNAME. Each P-I-D contains a list of identifiers, one for each plot associated with the selected GNAME. A typical P-I-D is shown in figure 6-16. The plot that is to be viewed/manipulated on-line is selected from this directory.
- d) Plot Transformation Menu (P-T-M) -- A list of parameters that can be modified such that the selected plot can be rotated and/or zoomed to create a new display (see fig. 6-17). Zooming a selected region of an on-line plot can also be performed by using the crosshairs provided by the Tektronix 4014.

The interactive-graphics execution mode is identified by the prompt character

I>

displayed at the beginning of each line on the terminal screen. At this point, a graphics execution directive is entered from the keyboard. The format of the user commands for making selections from the menus and directories, and for modifying the plot transformation parameters are summarized below.

<u>User Command</u>	<u>Applicable Menu</u>
FB,n	Function Menu
LP,n	G-D or P-I-D
Word1=x.x<,Word2=y.y>	Plot Transformation Menu (P-T-M)

The characters "FB" and "LP" are input as shown, in lieu of function buttons and a light pen which are associated with the more expensive, usually refreshable, graphics terminals. The integer that is input for "n" identifies a particular entry in the corresponding menu or directory previously selected by an "FB,n" command. A plot transformation parameter is input as a key-word followed by an equal sign and a decimal number, "x.x" or "y.y" that is to be assigned to the parameter.

Multiple commands can be concatenated by using semicolons as command delimiters. Each command or string of commands is terminated by depressing the CR (carriage return) key. Typical command formats are:

```
I>FB,4      CR
I>LP,3;FB,1  CR
I>FB,9;RY=60.,RX=45.,RZ=0.0;FB,1      CR
```

PLOT TRANSFORMATION MENU

ROTATION

1 RX	= 0.
2 RY	= 0.
3 RZ	= 0.
4 VX	= 1.
5 VY	= -1.
6 VZ	= 0.

ZOOM

7 XMIN	= 0.
8 XMAX	= 40.
9 YMIN	= 0.
10 YMAX	= 50.
11 ZMIN	= 0.
12 ZMAX	= 20.

ENTER KEY-WORD=XXX.XX TO RESET A PARAMETER (E.G., RY=45.0).

Figure 6-17. Plot Transformation Menu

PLOT TRANSFORMATION MENU

ROTATION

1 RX	= 0.
2 RY	= 0.
3 RZ	= 0.
4 VX	= 1. -1.
5 VY	= -1.
6 VZ	= 0. 1.

ZOOM

7 XMIN	= 0.
8 XMAX	= 40.
9 YMIN	= 0.
10 YMAX	= 50.
11 ZMIN	= 0.
12 ZMAX	= 20. 5.

ENTER KEY-WORD=XXX.XX TO RESET A PARAMETER (E.G., RY=45.0).

1> VX=-1.0,VZ=1.0,ZMAX=5.0:FB,1 CR

Figure 6-18. Example Use of the Plot Transformation Menu

To illustrate the user commands and a typical interactive dialog, assume the GNAMEs shown by figure 6-15 have been defined by previously executed GRAPHICS statements. Furthermore, assume that the displays for GNAME=GEOMETRY are identified by the P-I-D shown by figure 6-16.

Upon entering the interactive mode, the G-D menu is displayed on the screen (fig. 6-15). To view the plots associated with the GEOMETRY plot grouping, for this example, the user enters the command

```
I>LP,1;FB,1      CR
```

The selected GNAME is identified by a cross appearing on the screen automatically, followed by a display of the corresponding P-I-D (fig. 6-16).

All plots defined under the user-selected GNAME are identified in the P-I-D. To display one of these plots on the screen, a selection is made by the "LP,n" command. Thus, for example, if the user enters

```
I>LP,4;FB,1      CR
```

the selected plot title is identified by a cross on the screen followed by generation of the on-line plot.

A plot that has been generated on the screen can be transformed (rotated and/or zoomed) to create a new display of the same data. The function commands FB,8, FB,9 and FB,10 are used to perform transformations.

To display the Plot Transformation Menu, the following command is entered:

```
I>FB,8      CR
```

The current values of the orientation parameters and the X-Y-Z coordinate limits for the plot are shown in the P-T-M, as illustrated by figure 6-17.

To reset the values of the transformation parameters, the user may enter, for example, the following commands:

```
I>VX=-1.0,VZ=1.0,ZMAX=5.0;FB,1      CR
```

Updated values are displayed to the right of the original values, which are crossed out as shown by figure 6-18, and the new display is generated automatically on the screen.

For this example, a new viewing direction was selected in addition to zooming in on the Z-coordinate limits of the plot. It should be noted that the zoom parameters are effective in three dimensions. Thus, the original plot from which a zoomed display is generated can be recreated by resetting the values of the transformation parameters.

Transformation of a plot that has been generated on the screen can be performed without displaying the P-T-M. For example, the commands

```
I>FB,9;VX=-1.0,VZ=1.0,ZMAX=5.0;FB,1      CR
```

could be used instead of the two foregoing lines of commands to produce the same result.

The crosshairs provided by the Tektronix 4014 can be used to zoom a display shown on the screen. To activate the crosshairs, the following command is entered:

```
I>FB,10      CR
```

After the horizontal and vertical crosshairs are positioned to identify the zoomed area, the windowed portion of the display is then generated such that the zoomed area covers the screen. Figure 6-19 illustrates a display that is zoomed by use of the crosshairs.

When the interactive dialog is completed, execution control is returned to the Control Program by the following command:

```
I>FB,12      CR
```

The control statement immediately following the last executed GRAPHICS statement in the Control Program is then performed.

The FB,13 command allows the user to abort subsequent execution of the current ATLAS job. Generally, this option is used only when uncorrectable anomalies have been detected in the problem data by viewing the graphical displays on-line.

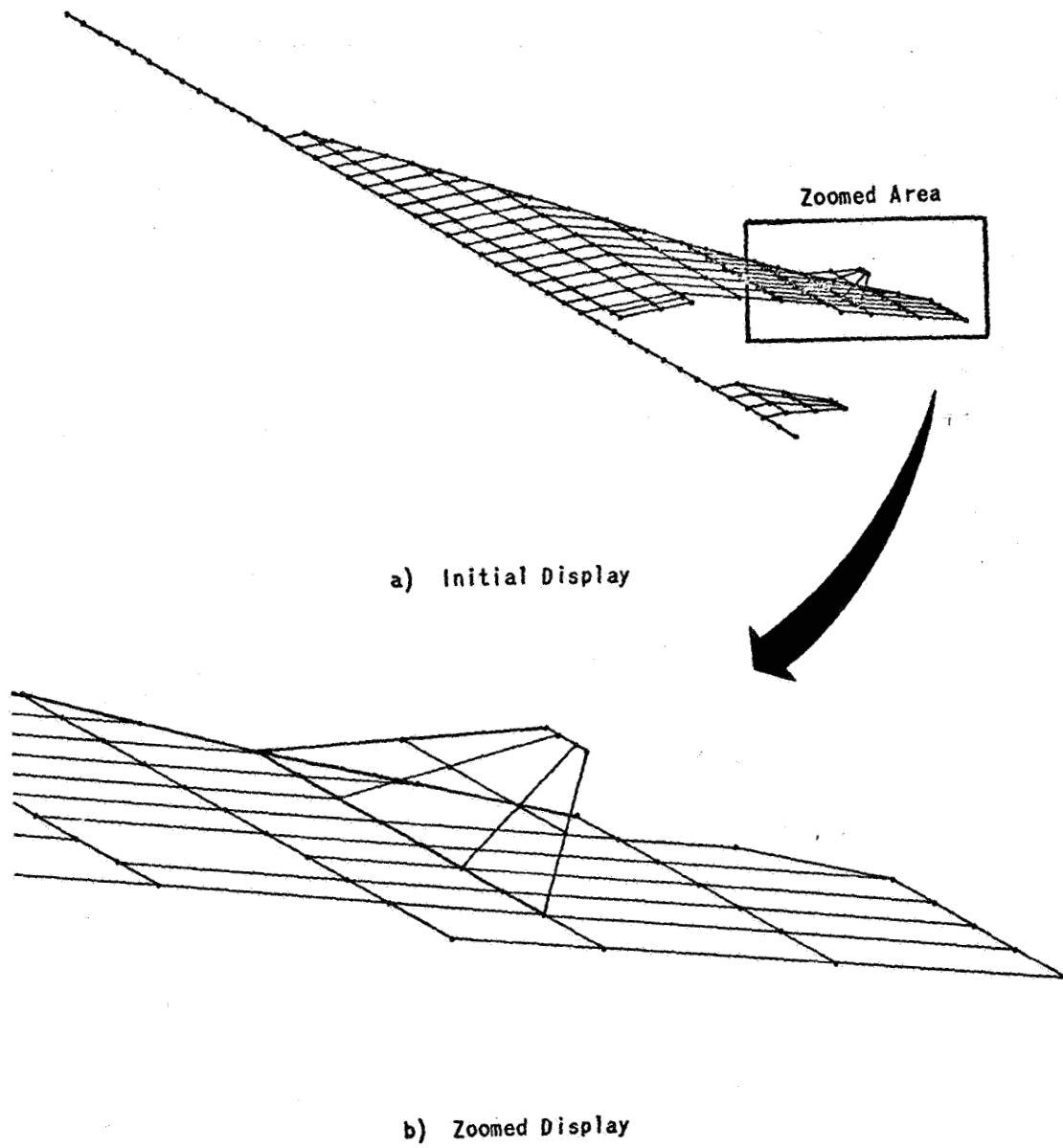


Figure 6-19. Zoomed On-line Display

6.4 INTERACTIVE PROCESSING (User's Manual; sec. 200)

Interactive Control-Program processing is initiated by use of the ATLAS control statement INTERACTIVE CONTROL as described in section 6.1. In this mode of system execution, the following activities can be performed using a remote computer terminal:

- a) Data and control files can be defined, interrogated and modified by edit capabilities inherent to the ATLAS INTERACTIVE CONTROL module.
- b) Control commands can be specified interactively to direct on-line execution of any ATLAS System module and to control data management functions.

All interactive execution directives are referenced as ATLAS control commands. The syntax of these commands is similar to that of the ATLAS execution-control statements described in section 6.1. Detailed descriptions are presented in section 200 of the User's Manual.

Each control command may be executed immediately after it is input by the terminal. Alternatively, multiple commands may be entered and stored to create a command procedure prior to execution thereof. A command procedure can call (initiate execution) another command procedure. They can be nested (concatenated) to form a maximum of 10 procedure levels.

Three modes of interactive processing are identified:

- a) ATLAS mode -- The basic mode for monitoring all interactive processing. This mode is entered either from the Control Program via an INTERACTIVE CONTROL statement or from a higher-level command procedure via execution of a RETURN or REVERT command.
- b) EDIT mode -- Entered from the ATLAS mode via the command &EDIT to create and edit either data files or execution-control files
- c) EXECUTE mode -- Entered from the ATLAS mode via the "&" execute symbol. Execution of all control commands is monitored by this mode.

The ATLAS mode is re-entered upon completion of activities performed in either the EDIT or the EXECUTE mode. The functions of the processing modes, as illustrated in figure 6-20, are described below.

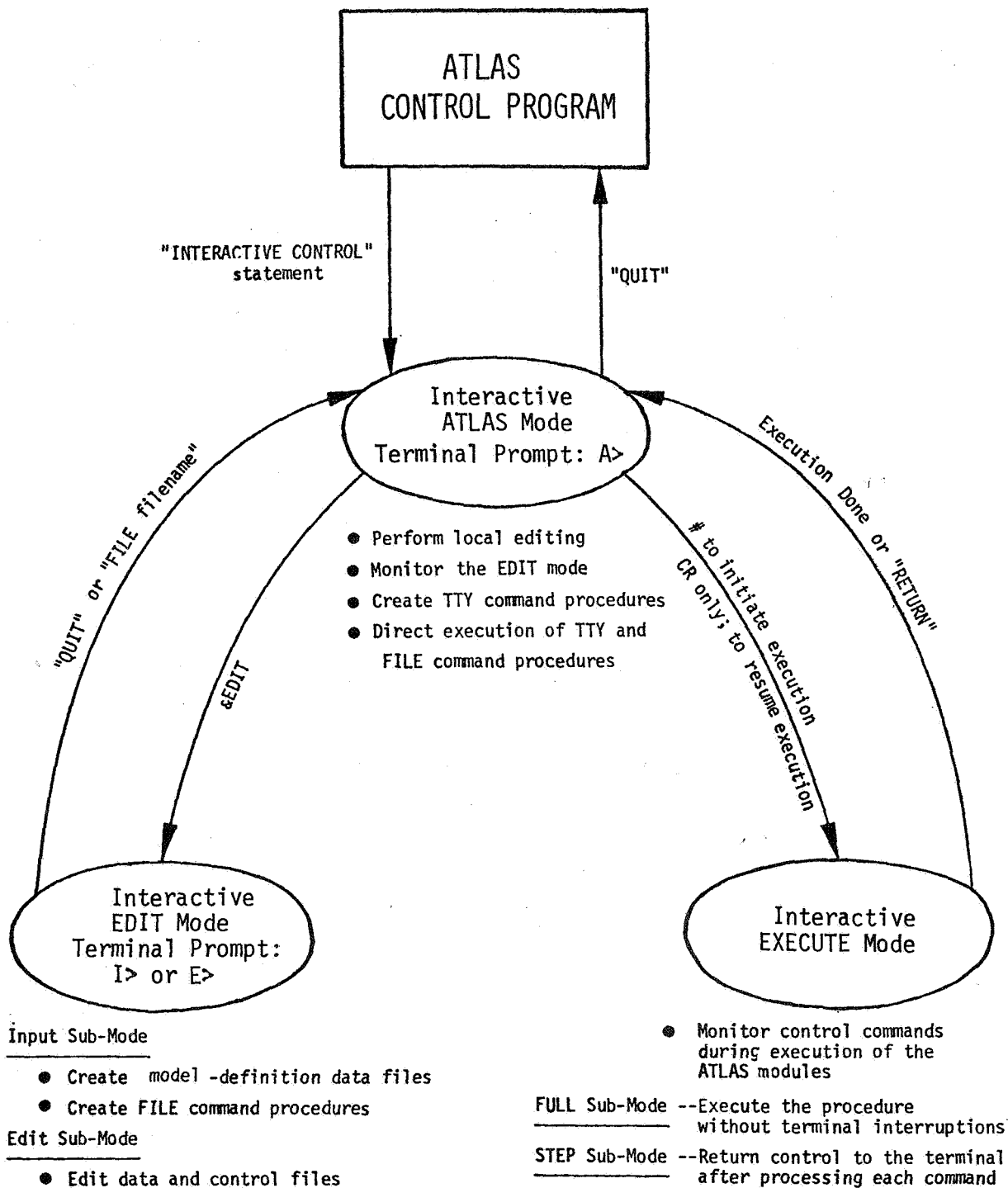


Figure 6-20. Interactive Execution Control and Processing Functions

6.4.1 ATLAS Interactive Mode

All interactive processing is monitored from the ATLAS mode. This mode is identified by the prompt character

A>

displayed at the beginning of each line on the terminal screen. At this point, an ATLAS command is entered using the keyboard. If the command is to be executed immediately, the execute symbol "#" should be typed, followed by depressing the CR (carriage return) key. If the command is to be stored for subsequent execution, the symbol "#" should not be input. Multiple commands entered without the "#" symbol define a TTY command procedure.

Entry of the execute symbol "#" initiates processing of a command procedure. During execution of the procedure, control activity may be reverted to the ATLAS mode from the EXECUTE mode at user-selected interrupt points within the procedure.

Special "priority" commands may be entered in the ATLAS mode to change the central-memory field length, to enter the EDIT mode, or to edit the last "n" lines entered by the keyboard. Priority commands are processed immediately followed by the ATLAS mode being resumed for further processing of control commands.

6.4.2 EDIT Interactive Mode

The EDIT mode is entered from the ATLAS mode by input of the "&EDIT" priority command. The editor operates in either the input sub-mode or the edit sub-mode identified by the prompt characters

I> or E>

The edit capabilities, as summarized in table 6-5, are described in detail in appendix H of the User's Manual.

The EDIT program can be used to edit files of line images. Each file may contain any number of logical records. Each logical record may contain any number of line images and each line may contain 1 to 140 characters. The program uses a system of requests to locate a specific line and/or to perform a desired modification, insertion, or deletion. Each EDIT request consists of the request name, followed by parameters as needed. Specific lines in a file are referenced by means of a pointer. The pointer may be moved by execution of specific requests and is used by other requests. The pointer is not visible to the user, but he can easily determine its position in the file at any time by using the PRINT request.

Table 6-5. Summary of the ATLAS Text-Editor Requests

<u>Environment Selection:</u>	
INPUT (type a (C))	-enter input mode
QUIT	-return from input mode
	-leave EDIT
<u>Message Mode Selection:</u>	
BRIEF a b	-BRIEF mode
VERIFY n	-VERIFY mode
<u>Pointer Movement:</u>	
BOTTOM	-go to bottom of file
FIND line	-search for line
LOCATE :string1:string2:...	-search for a string
NEXT n	-advance n lines
NEXT :string1:string2:...	-advance to a string
TOP	-go to top of file
UP n	-go up n lines
UP :string1:string2:...	-go up to a string
<u>Modification of Line Images:</u>	
AFTER :string1:string2: n m	-insert string2 after string1
BEFORE :string1:string2: n m	-insert string2 before string1
BLANK mask	-blank out characters
CHANGE :string1:string2: n m	-change string1 to string2
DELETE n	-delete n lines
DELETE :string1:string2:...	-delete until a string is found
INSERT line	-insert "line" as line image
OVERLAY line	-overlay characters
REPLACE line	-replace line image with "line"
<u>File Handling:</u>	
FILE filename un pw	-save file and leave EDIT
SAVE filename un pw	-save file and return to input mode
<u>Information Request:</u>	
PRINT n	-print n lines
PRINT :string1:string2:...	-print to a string
<u>Special Character and Format Conventions:</u>	
LINEND character	-change end-of-line character
TABDEF character	-change tab character
TABSET n1 n2 ... nn	-change tab settings
WIDTH n	-specify line length
ZONE a b	-specify 1st and last columns
<u>Block Operations:</u>	
START	-define start of block
END	-define end of block
DUP	-duplicate block
MOVE	-move block
REMOVE	-delete block
<u>Miscellaneous:</u>	
REPEAT n	-repeat BLANK and OVERLAY requests
X /request1/request2/...	-save requests
X n	-execute saved requests n times
Y /request1/request2/...	-same as X
Y n	-same as X

Command procedures created in the EDIT mode are referenced as FILE procedures. An image of the commands is stored in a user-named file which is subsequently processed in the ATLAS mode.

6.4.3 EXECUTE Interactive Mode

Processing of those control commands that initiate execution of other ATLAS modules is monitored by the EXECUTE interactive mode. On-line interaction during the execution of a module is only provided during interactive graphics as described in section 6.3.

Execution of a command procedure is performed in either the FULL or STEP sub-mode. In the FULL mode, all commands in the procedure are processed without any interruption allowed from the terminal except that provided by a RETURN command included in the procedure. In the STEP mode, control returns to the terminal after processing of each command. At this point, any type of further action may be initiated. A new command procedure, for example, may be processed prior to resuming operations on the current procedure.

6.4.4 Example Interactive Processing Session

The following example illustrates an ATLAS interactive session where the basic model geometry is preprocessed and checked for errors using the interactive graphics capability. Figure 6-21 shows the model-definition data deck used for this example.

The detected errors are corrected using the ATLAS edit capability and a plot of the final model is obtained. The comments enclosed within braces { } to describe the interactive processing do not appear on the terminal screen during an interactive session.

```

*/ MODE2 /
BEGIN NODAL DATA
CYL BODY 100. 0. 100. 100. 0. 150. 150. 0. 100.
1002 50. 0. 0. TO 1018 50. 180. 0.
**4 1000 0. 0. 15 0 1000 0. 0. 15.
RESUME GLOBAL
1010 100. 50. 100. 5. TO 1130 150. 170. 110. 3. BY 10
**4 1000 15. 0. 0. -.2 0 1000 7.5 0. 0. -.1 0 0
END NODAL DATA
BEGIN STIFFNESS DATA
BEGIN ELEMENT DATA
BEAM 1002 1003 .5 .1 .1 TO 1017 1018
**4 0 1000 1000 0. 0. 0. 0 1000 1000
ROD 1002 2002 .2 TO 1018 2018
**3 0 1000 1000 0. 0 1000 1000
PLATE 1002 1003 2003 2002 .02 TO 1017 1018 2018 2017
**3 0 1000 1000 1000 1000 0. 0 1000 *=3
SPAR 1010 1020 .1 2. TO 1120 1130 BY 10 10
**4 0 1000 1000 0. 0. 0 1000 1000 0 0 0
COVER 1010 1020 2020 2010 .06 .05 TO 1120 1130 2130 2120 BY +
10 *=3
**3 0 1000 *=3 -.005 -.005 0 1000 *=3 0 0 0 0 0
END ELEMENT DATA
END STIFFNESS DATA
END PROBLEM DATA

```

Figure 6.21. Data Deck for Example Interactive Session

{Log on}

78/05/03. 08.51 45.
EKS2 175D N0452.09E 78/05/01.DS-0 03.08.24. 78/05/03
USER NUMBER {Input your account number and pass-word here}

TERMINAL 10. TTY
RECOVER/USER ID {Input your name here}

{Get the ATLAS system files}

N>GET,CATM/UN=UATLASU

N>CALL,CATM,ATTACH

08.52.19.*****
08.52.21.* ATLAS VERSION 4.0 *
08.52.22.* PRODUCTION VERSION *
08.52.23.*****
COMMENT.END FILES

{Get the file containing the ATLAS data deck-fig. 6-21}

N>GET,SGX

{Get an absolute version of a Control Program
created for the interactive execution mode. This
Control Program contains the INTERACTIVE CONTROL
statement.}

N>GET,CONTROL=SGCONTA

{Switch to the KIT batch mode, increase the
field length to 130000 octal, and initiate
ATLAS system execution.}

N>BATCH,130000

C>ATLAS,SGX,OUTB,OUTPUT

EXECUTING CONTROL CP SECS .103

{Preprocess the ATLAS data deck}

A>READ INPUT\$#

EXECUTING INPTOLF CP SECS .124

EXECUTING CONTROL CP SECS 1.446

+++ WARNING - PREVIOUS MODULE INCREASED WARNING COUNT BY 17

{ In order to plot the model to identify any problems,
SUBSET DEFINITION data must be added to the data deck.
The ATLAS edit capability is used to create a file
(SUBFIL) that contains the SUBSET DEFINITION input data
required to define the plots desired. }

```
A>&EDIT,SUBFIL
NEW FILE
I>BEGIN SUBSET DEFINITION
I>SUBSETS OF STIFFNESS SET 1
I>N1 = ALL
I>E1 = ALL
I>END SUBSET DEFINITION
I>END PROBLEM DATA
I>
E>FILE
/* 075100 OCTAL REQUIRED TO EDIT
```

{Preprocess the SUBSET DEFINITION input data}

```
A>READ INPUT(I=SUBFIL,MODE2)$#
EXECUTING INPTOLF      CP SECS      1.487
EXECUTING CONTROL      CP SECS      1.520
```

{ Execute the EXTRACT postprocessor to extract the
nodes and elements to be plotted. }

```
A>EXECUTE EXTRACT(EXNAME=GEOM,LSUB=KGRID,NSUB=N1,ESUB=E1)$#
EXECUTING EXTROLF      CP SECS      1.550
EXECUTING CONTROL      CP SECS      2.368
```

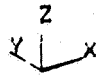
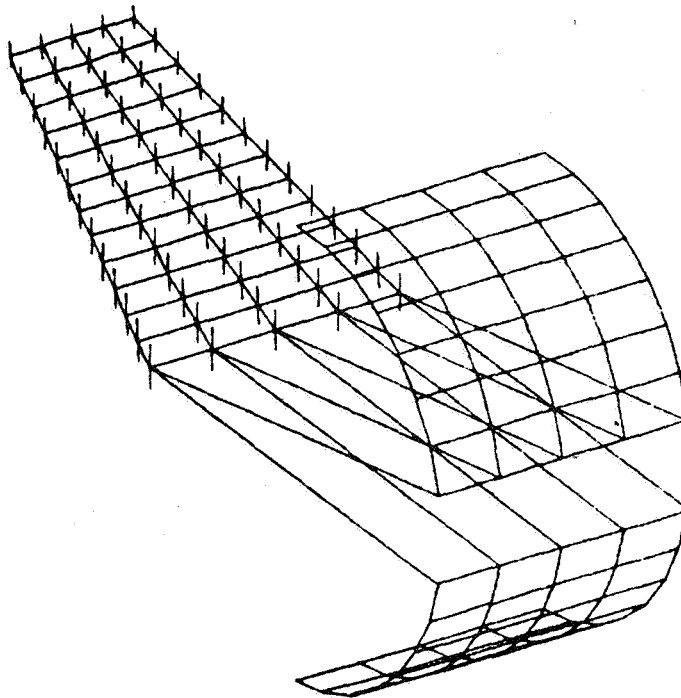
{ The GRAPHICS postprocessor is executed to display the
model geometry - fig. 6-22 }

```
A>EXECUTE GRAPHICS(GNAME=GEOM,TYPE=ORTH,EXNAME=GEOM)$#
```

{ The plot indicates that the definition of the
body cylindrical coordinate system is in error.
The axes are oriented incorrectly. Using the ATLAS edit
capability, the coordinate system definition
is corrected and saved on the data file SGX. }

GNAME • GEOM

PLOT ID • GEOM . GEOMETRY



RZ • 125
RY • 25
RX • 0 0

OFFLINE
SCALE • 1/38 86
SCALE-X • 1/45 49
SCALE-Y • 1/58 01
SCALE-Z • 1/42 88

ONLINE
SCALE • 1/38 86
SCALE-X • 1/45 49
SCALE-Y • 1/58 01
SCALE-Z • 1/42 88

EXNAME • GEOM KSET • 1 NSUB • N001
ESUB • E001 LSUB • KGRID

Figure 6-22. On-line Display of Initial Model Geometry

A>&EDIT,SGX

E>N 3

CYL BODY 100. 0. 100. 100. 0. 150. 150. 0. 100.

E>C /150. 150. / 150. 50. /

CYL BODY 100. 0. 100. 100. 0. 150. 50. 0. 100.

E>N 2

*+4 1000 0. 0. 15. 0 1000 0. 0. 15.

E>C /15./-15./ 1 *

*+4 1000 0. 0. -15. 0 1000 0. 0. -15.

E>FILE

/* 075100 OCTAL REQUIRED TO EDIT

{The random access files containing the incorrect
data are purged, and the corrected data file
is preprocessed.}

A>PURGE FILES (DATARNF,EXTRNRF) \$#

A>READ INPUT\$#

EXECUTING INPTOLF CP SECS 3.301

EXECUTING CONTROL CP SECS 4.598

+++ WARNING - PREVIOUS MODULE INCREASED WARNING COUNT BY 9

{Check the file containing the SUBSET DEFINITION
input data.}

A>&EDIT,SUBFIL

E>P *

BEGIN SUBSET DEFINITION

SUBSETS OF STIFFNESS SET 1

N1 = ALL

E1 = ALL

END SUBSET DEFINITION

END PROBLEM DATA

EOI

E>FILE

/* 075100 OCTAL REQUIRED TO EDIT

{Preprocess the SUBSET DEFINITION input data}

A>READ INPUT(I=SUBFIL,MODE2) \$#

EXECUTING INPTOLF CP SECS 4.639

EXECUTING CONTROL CP SECS 4.674

{Execute the EXTRACT postprocessor to identify
the nodes and elements to be plotted.}

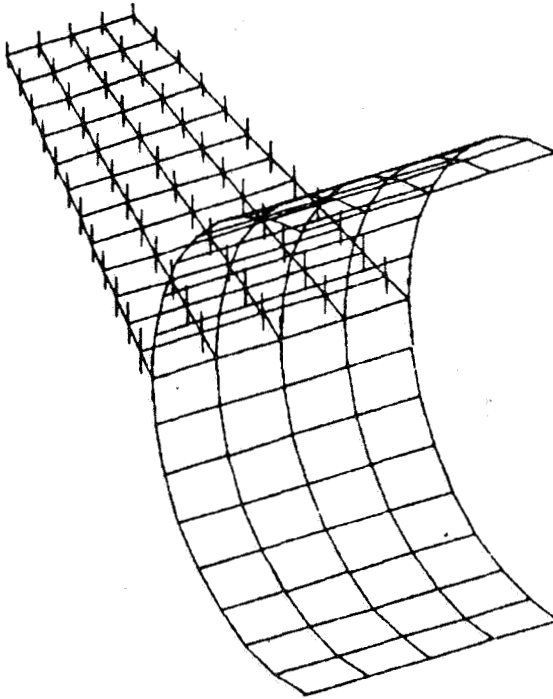
```
A>EXECUTE EXTRACT (EXNAME=NEWGEOM,LSUB=KGRID,NSUB=N1,ESUB=E1) $#
EXECUTING EXTROLF      CP SECS      4.707
EXECUTING CONTROL      CP SECS      5.554
```

{ The GRAPHICS postprocessor is executed to display the
corrected model geometry-fig. 6-23. }

```
A>EXECUTE GRAPHICS (GNAME=NEWGEOM,TYPE=ORTH,EXNAME=NEWGEOM) $#
```

GNAME = NEWGEOM

PLOT ID = NEWGEOM. GEOMETRY



RZ = 125
RY = 25
RX = 0 0

OFFLINE
SCALE = 1/37 87
SCALE-X = 1/44 33
SCALE-Y = 1/56 52
SCALE-Z = 1/41 78

ONLINE
SCALE = 1/37 87
SCALE-X = 1/44 33
SCALE-Y = 1/56 52
SCALE-Z = 1/41 78

```
EXNAME = NEWGEOM  KSET = 1      NSUB = N001
ESUB = E001      LSUB = KGRID
```

Figure 6-23. On-line Display of Corrected Model Geometry

{ The data and model are correct; save the
preprocessed data for later restart. }

A>SAVE FILES##

EXECUTING OTPTOLF	CP SECS	6.453
EXECUTING CONTROL	CP SECS	6.714

{ Terminate the ATLAS interactive processing
session, and return to the on-line
execution of the Control Program. }

A>QUIT\$#

EXECUTING GRAPOLF	CP SECS	6.736
EXECUTING CONTROL	CP SECS	6.752
6.661 CP SECONDS EXECUTION TIME		

C>

7.0 JOB CONTROL CARDS

The CDC 6600/CYBER computer system control cards (job control deck) required for execution of an ATLAS job are described in this section. The example control cards used for illustration purposes are valid for the NOS 1.1, NOS 1.2, and the KRONOS 2.1 computer operating systems. Because some of the control cards are dependent on the operating system and hardware configuration of the particular CDC installation, variations in the illustrated card formats must be accommodated by the user, as necessary.

The job control decks required for most types of ATLAS jobs are described in the following sections. Each control card statement begins in column one of a card. The control card deck is placed in front of the ATLAS execution-control deck, as illustrated previously in figure 4-1, prior to submittal of the job for batch execution. When executing in the on-line or interactive modes, the job control card statements are input directly from the terminal keyboard.

The functions performed by the basic control cards used to execute most ATLAS jobs can be grouped as follows:

- a) Identify the CDC job and the user account, specify the estimated computer resource requirements (the Job Card and Account Card).
- b) Provide access to the ATLAS executable codes as stored on the CDC permanent files (the "GET,CATM" and the "CALL(CATM,ATTACH)" statements).
- c) Access the user-defined ATLAS execution-control deck (source format), translate its user-oriented languages into a computer-sensible language (absolute format), and load (enter) the resulting Control Program for execution (the "CALL(CATM,CONTROL)" statement).
- d) Execute ATLAS
- e) Save or load (restore) the ATLAS data base for execution restart of sequential ATLAS jobs.
- f) Create a plot data file for generating off-line graphical displays.

ATLAS is designed to utilize computer core dynamically so that at execution time, all of the core memory that is made available by the field-length estimate is generally used. The minimum core requirement for ATLAS is approximately 110000 octal words.

Depending on the size and type of problem being executed, this minimum must be adjusted according to the resource requirement table presented in section 11 of the User's Manual.

7.1 EXAMPLE JOB CONTROL DECKS

The example job control decks presented below illustrate many of the typical situations encountered during execution of ATLAS. Further descriptions of control card options are presented in section 11 of the User's Manual.

7.1.1 Batch Execution -- Model and Execution Data Decks on Cards

Job Card

Account Card

```
GET (CATM/UN=UATLASU)
CALL (CATM,ATTACH)
CALL (CATM,CONTROL)
ATLAS (PL=100000)
```

7.1.2 Batch Execution -- Model and Execution Data Decks in Permanent Files

Job Card

Account Card

```
GET (CATM/UN=UATLASU)
CALL (CATM,ATTACH)
GET (cpfile)
CALL (CATM,CONTROL (INPUT=cpfile))
GET (atdata)
ATLAS (PL=100000,atdata)
```

In this example, the ATLAS model-definition data were previously defined and saved in the "atdata" file, whereas the ATLAS execution-control deck was previously saved in the "cpfile" file.

7.1.3 On-line Execution -- Model and Execution Data Decks in Permanent Files

Log On Statements

BATCH,field length.

```
GET (CATM/UN=UATLASU)
CALL (CATM,ATTACH)
GET (cpfile)
CALL (CATM,CONTROL (INPUT=cpfile,OUTPUT=out1))
GET (atdata)
ATLAS (PL=100000,atdata,out2,OUTPUT)
```

In this example, printout from the compilation of the ATLAS execution-control deck is directed to the "out1" file. During

the execution of ATLAS, all computer printout is directed to the "out2" file, and a copy of the DAYFILE messages is directed to the OUTPUT file (i.e., the terminal screen).

7.1.4 Using a Control Program in Absolute Format

When a number of ATLAS jobs are to be executed for solving similar types of problems, the Control Program can remain unchanged. A particular sequence of ATLAS execution-control statements for performing a stress analysis, for example, can be used any number of times. To avoid repeated translation of the user's executive-control source statements into absolute format (a computer-sensible language) during each ATLAS job, the absolute version of the Control Program can be stored in permanent file by using the following control cards directly after the "CALL(CATM,CONTROL)" statement:

```
SAVE(CONTROL=mycp)
```

In the subsequent ATLAS jobs, use of the absolute version of the Control Program stored in the "mycp" file is illustrated by the following job control deck.

```
Job Card
Account Card
GET(CATM/UN=UATLASU)
CALL(CATM,ATTACH)
GET(CONTROL=mycp)
ATLAS(PL=100000)
```

7.1.5 Job Execution Checkpoint

The ATLAS input data and calculated data can be saved for restart of problem analysis by a subsequent ATLAS job. Data to be saved are identified by the SAVE execution-control statements (see sec. 6). The total ATLAS data base or selected blocks of data residing in the ATLAS data base can optionally be saved at multiple points within the sequence of execution-control statements. Checkpoints can only be established between executions of ATLAS modules. Intramodule checkpoints are not provided.

Selected checkpoint data for small jobs can be stored in a permanent file by using the following control card after the "ATLAS" job control card.

```
SAVE(Savefile=job1)
```

The name of "Savefile" must be one of the ATLAS restart file names as identified by the corresponding SAVE execution-control

statement(s). The selected data are stored in the user-assigned "job1" permanent file.

For a large job, the selected checkpoint data should be saved on magnetic tape by using the following control cards after the "ATLAS" job control card.

```
REQUEST(TAPE,F=I, LB=KL, PO=AW) SAVE
REWIND(Savefile)
COPY(Savefile,TAPE)
RETURN(TAPE)
```

7.1.6 Job Execution Restart

To restart a job execution using previously-saved checkpoint data, the following control cards should be used before the "ATLAS" job control card.

For data saved on permanent file, use:

```
GET(Savefile=job1)
```

For data saved on magnetic tape, use:

```
REQUEST(TAPE,VSN=xxxxxx,F=I, LB=KL)
COPY(TAPE,Savefile)
REWIND(Savefile)
RETURN(TAPE)
```

These statements are the counterpart to those statements described in section 7.1.5 for saving the checkpoint data. Similarly, the name of "Savefile" must be one of the ATLAS restart file names as identified by the corresponding LOAD execution-control statement(s) used for the new job.

The following job control deck illustrates a job restart, as well as establishing another job execution checkpoint. The ATLAS restart files SAVESS1 and SAVESS2 are used in this example:

```
Job Card
Account Card
GET(CATM/UN=UATLASU)
CALL(CATM,ATTACH)
CALL(CATM,CONTROL)
GET(SAVESS1=job1)
ATLAS(PL=100000)
SAVE(SAVESS2=job2)
```

7.1.7 Plot Data File Creation

Whenever off-line plots are requested by an EXECUTE GRAPHICS statement, the following control card, followed directly by appropriate comments for the selected plot device operator, should be used after the "ATLAS" job control card.

PLOTFIL(Plotter,TAPE99,0)

The word "Plotter" must be input as one of the following words to identify which plot hardware unit is to be used.

SC4020 -- the Stromberg Carlson SC4020 microfilm plotter or
 the COMP-80 photocomposer

CALCOMP -- the CALCOMP Model 763

GERBER -- the GERBER drafting machine or the Versatec
 electrostatic quick-plot unit

Only one plot hardware system may be requested per ATLAS job. The particular unit associated with a job is that identified by the first EXECUTE GRAPHICS statement in the ATLAS execution-control deck. All plots created by an ATLAS job can be generated using any one of the foregoing plot units with equal ease, depending on the required plot size, accuracy, and quality.

8.0 EXAMPLE PROBLEMS

Examples of the following types of analyses, as performed using the ATLAS System, are presented in this section.

- a) Stress
- b) Bifurcation buckling
- c) Weights
- d) Vibration
- e) Flutter
- f) Substructured stress analysis
- g) Substructured vibration analysis
- h) Structural resizing

Except as noted, each example problem uses the same structural model as shown in figures 8-1 through 8-3. Figure 8-2 illustrates a layout of the model that was created prior to setting up the model-definition input data. Use of the ATLAS GRAPHICS postprocessor to plot the model data, as shown in figure 8-3, offers convenient methods of verifying the structural model prior to performing any problem solutions. These figures illustrate the types of "road maps" that should be developed during the process of structural idealization for orderly management of the data during problem solving.

The model is comprised of mid-surface nodes and SPAR and COVER elements. The bracing struts are BEAM elements. The wing box width and depth vary linearly from root to tip. All SPAR elements have flange areas of 1.5 and web thicknesses of 0.25. All COVER elements have surface thicknesses of 0.25. The BEAM elements have areas of 0.25, bending moments of inertia of 0.03 and torsional constants of 0.06. All nodes have the local reference frame SPAN as their analysis frame.

The following items are presented and discussed for each example problem:

- a) Additional components of model
- b) Analysis performed
- c) Execution-control data
- d) Model-definition data
- e) Results

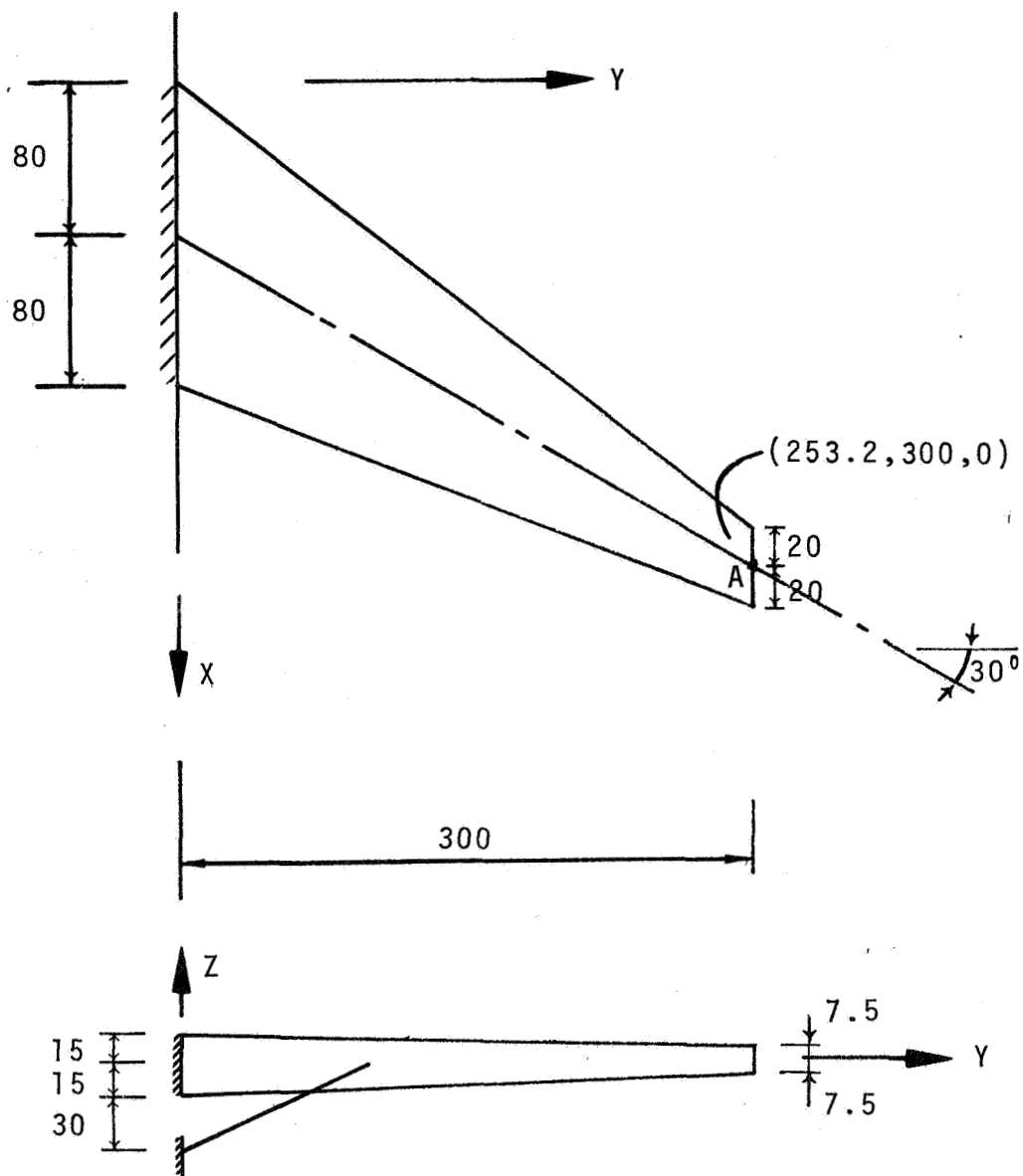


Figure 8-1. Geometry of the Wing Box Structure

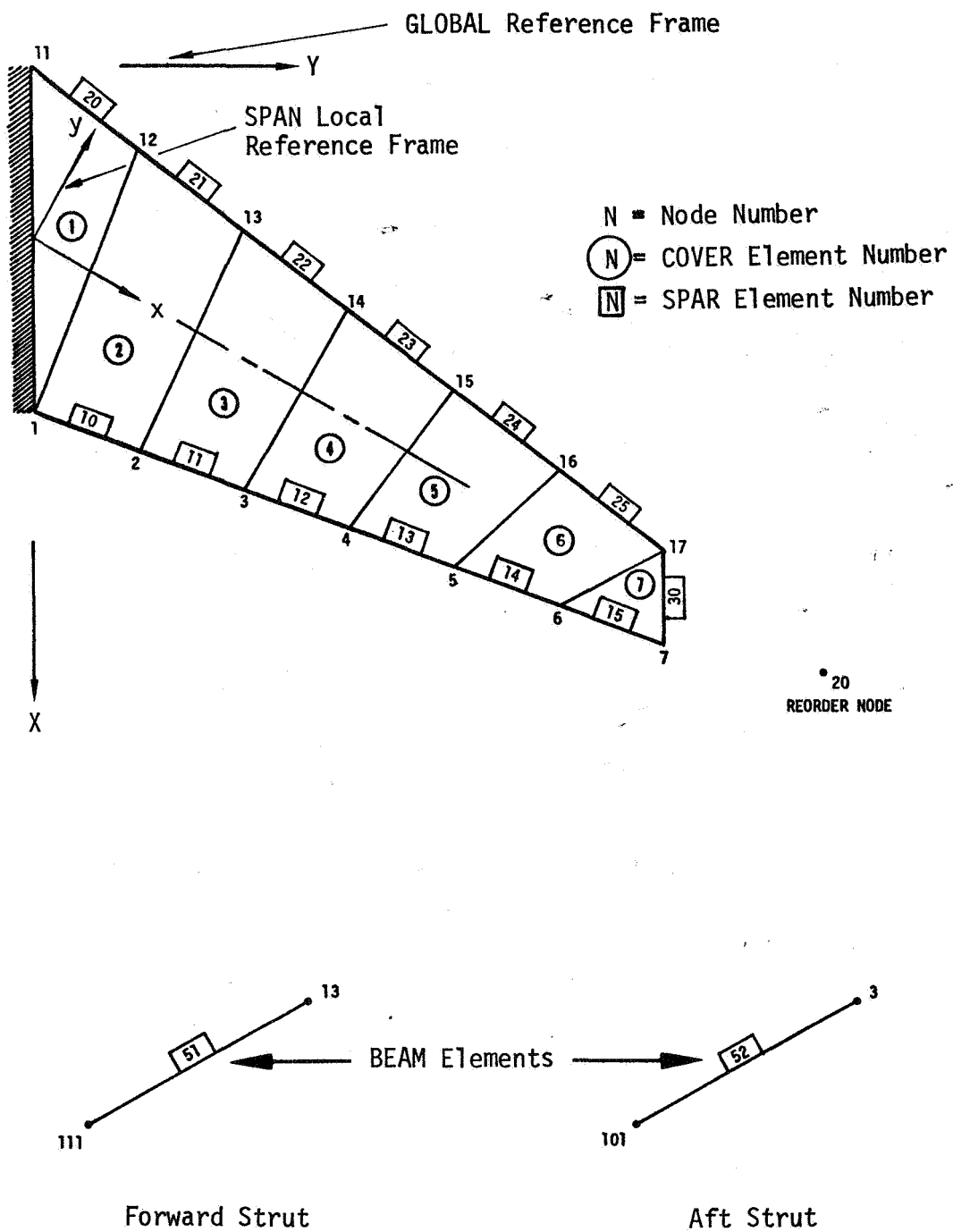


Figure 8-2. Layout of the Structural Model

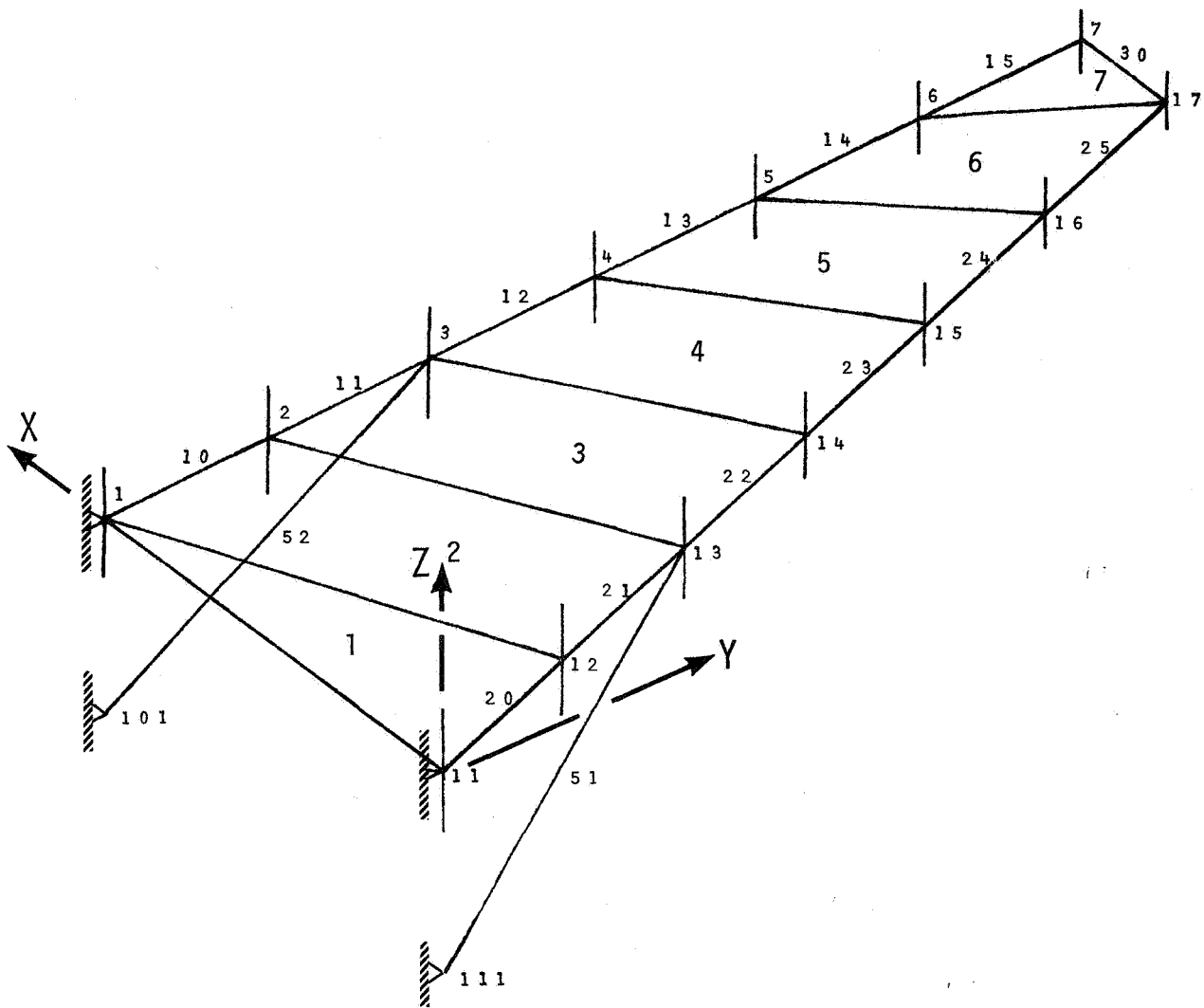


Figure 8-3. Wing Box Structural Model as Plotted Using the GRAPHICS Postprocessor

8.1 EXAMPLE 1: STRESS ANALYSIS

8.1.1 Description of Analysis

This example consists of a stress analysis of the wing box described in section 8.0. The box is fully supported at the root and is subjected to three loading conditions:

- a) A distributed pressure load acting in the GLOBAL Z direction. The pressure is uniform over each element and varies from a maximum at the root to a minimum at the tip.
- b) A concentrated tip load acting in the GLOBAL Z direction and divided equally between front and rear spars.
- c) A thermal loading consisting of a uniform ΔT of 200°F on the upper surface and -200°F on the lower surface.

8.1.2 Execution-Control Data

1	BEGIN CONTROL PROGRAM STRESS
2	PROBLEM ID (EXAMPLE STRESS ANALYSIS)
3	READ INPUT (MODE2)
4	PRINT INPUT (NODAL)
5	PRINT INPUT (STIFFNESS)
6	PRINT INPUT (LOADS)
7	PERFORM STRESS
8	PRINT INPUT (BC)
9	PRINT OUTPUT (LOADS)
10	PRINT OUTPUT (DISP)
11	PRINT OUTPUT (STRESS)
12	PRINT OUTPUT (REACTIONS,EQCHK)
13	END CONTROL PROGRAM

NOTES:

- ① Required first statement in an ATLAS execution-control deck. The name selected for this Control Program is STRESS.
- ② Assigns a title to the problem. This title will appear in the printout.
- ③ Directs ATLAS to read and check the model-definition data deck.

- ④-⑥ Request formatted printout of the input data for:
- ④ nodes
 - ⑤ stiffness elements
 - ⑥ loads
- ⑦ Directs ATLAS to perform a stress analysis. The STRESS execution-control procedure calculates stiffnesses, forms the equilibrium equations, and solves for displacements, stresses, and reactions.
- ⑧ Requests a formatted printout of the input boundary condition data. Use of this statement after execution of the STIFFNESS processor (as included in the STRESS procedure in statement 7) results in inactive freedoms (those with zero stiffness) being identified in the printout.
- ⑨-⑫ Request formatted printout of the calculated data for:
- ⑨ the cumulated equivalent nodal loads and resultants of applied loads
 - ⑩ nodal displacements
 - ⑪ element stresses
 - ⑫ reactions and equilibrium check (sum of the applied loads and the reactions)
- ⑬ Required last statement in an ATLAS execution-control deck

8.1.3 Model-Definition Data

```

1 BEGIN NODAL DATA
2 REC SPAN 80. 0. 0. 253.2 300. 0. 80. 0. 100.
3 20 1000. 0. 0.
4 RESUME GLOBAL
5 ANALYSIS FRAME SPAN
6 1 160. 0. 0. 15. TO 7 273.2 300. 0. 7.5
7 11 0. 0. 0. 15. TO 17 233.2 300. 0. 7.5
8 101 160. 0. -45.
9 111 0. 0. -45.
10 REORDER FROM 20
11 END NODAL DATA

12 BEGIN STIFFNESS DATA
13 BEGIN ELEMENT DATA
14 SPAR N10 1 2 .25 1.5 TO N15 6 7
15 SPAR N20 11 12 .25 1.5 TO N25 16 17
16 SPAR N30 7 17 .25 1.5
17 COVER N1 1 12 11 .25

```

```

18      COVER N2      1      2      13      12 .25      TO      N6      5      6      17      16
19      COVER N7      6      7      17      .25
20      BEAM  N51     13     111.25  0.  0.      .06  .03  .03
21      BEAM  N52      3     101      **
22      END ELEMENT DATA
23      END STIFFNESS DATA

24      BEGIN BC DATA
25          SUPPORT ALL FOR 1      11      101      111
26      END BC DATA

27      BEGIN LOADS DATA
28          LOAD CASE ID AIRLOAD **ELEMENT PRESSURE LOADING#
29          LOAD CASE ID TIPLOAD **2000 LB. LOAD AT TIP#
30          LOAD CASE ID THERMAL **+200 DEG. UPR. AND -200 ON LWR. #
31      BEGIN NODAL LOAD DATA
32          CASE TIPLOAD
33              7      17      FZ      -1000.
34      END NODAL LOAD DATA
35      BEGIN ELEMENT LOAD DATA
36          DIRECTION GLOBAL 0.  0.  1.
37          CASE AIRLOAD
38              1      3.5
39              **6 1      -.5
40      END ELEMENT LOAD DATA
41      BEGIN ELEMENT THERMAL LOAD DATA
42          CASE THERMAL
43              10 TO 15, 20 TO 25, 30 200. -200. 200. -200.
44              1 TO 7 200. -200.
45      END ELEMENT THERMAL LOAD DATA
46      END LOADS DATA

47      END PROBLEM DATA

```

NOTES:

- ① Identifies the start of the NODAL data set
- ② Defines a local rectangular reference frame with the name SPAN
- ③ Defines node 20 with x-y-z coordinates (100.0,0.,0.) measured relative to the SPAN coordinate system.
- ④ Identifies the GLOBAL reference frame as the coordinate system used for subsequent nodal input coordinates
- ⑤ Identifies reference frame SPAN as the analysis frame for subsequent nodes

- ⑥ Defines mid-surface nodes 1,2,...,7 along the rear spar
- ⑦ Defines mid-surface nodes 11,12,...,17 along the front spar
- ⑧,⑨ Define strut nodes 101 and 111
- ⑩ Specifies nodal data to be reordered, based on distances from node 20, so that the stiffness matrix bandwidth and solution costs are minimized
- ⑪ Identifies the end of the NODAL data set
- ⑫ Identifies the start of the STIFFNESS data set
- ⑬ Identifies the start of the structural element data subset
- ⑭,⑮ Define the rear and front spar elements; element numbers 10-15 and 20-25, respectively.
- ⑯ Defines the wing tip closure rib (element number 30)
- ⑰,⑱ Define the wing surface skin (element numbers 1-7)
- ⑲,⑳ Define the wing struts (element numbers 51 and 52)
- ㉑ Identifies the end of the structural element data
- ㉒ Identifies the end of the STIFFNESS data set
- ㉓ Identifies the start of the boundary condition (BC) data set
- ㉔ Defines all kinematic freedoms for nodes 1,11,101, and 111 to be supported (fixed)
- ㉕ Identifies the end of the (BC) boundary condition data set
- ㉖ Identifies the start of the LOADS data set
- ㉗,㉘ Define three loadcase labels and their alphanumeric titles
- ㉙ Identifies the start of the NODAL-LOAD data subset
- ㉚,㉛ Define concentrated loads in the GLOBAL direction at nodes 7 and 17 for loadcase TIPLOAD

- ③④ Identifies the end of the NODAL-LOAD data subset
- ③⑤ Identifies the start of an element-distributed LOAD data subset
- ③⑥ Specifies load-direction cosines relative to the GLOBAL reference frame
- ③⑦ Identifies AIRLOAD as the loadcase label for subsequent element loads
- ③⑧ Specifies a uniform pressure intensity of 3.5 on element number 1
- ③⑨ Generates uniform pressure loads for elements 2-7; the pressure intensity on element 7 is 0.5
- ④① Identifies the end of the ELEMENT LOADS data
- ④② Identifies the start of an ELEMENT THERMAL load data subset
- ④③ Identifies THERMAL as the loadcase label for subsequent loads
- ④③④④ Define a temperature increment of 200°F for the upper surface of the wing finite elements and -200°F for the lower surface elements
- ④⑤ Identifies the end of the ELEMENT THERMAL LOADS data
- ④⑥ Identifies the end of the LOADS data set
- ④⑦ Identifies the end of the model-definition data deck

8.1.4 Results

Representative output from the stress analysis is shown on the following pages. Printout of input data (nodal, stiffness, boundary condition, and loads) and output data (loads, displacements, stresses, and reactions) are included.

***** NODAL DATA *****
 PROBLEM ID -- EXAMPLE STRESS ANALYSIS DATA SET NUMBER = 1

USER NODE	INPUT RECORD	INTERNAL NODE	X COORD	Y COORD	Z COORD	DX COORD	DY COORD	DZ COORD	NODAL REF. FRAME	ANALYSIS REF. FRAME
1	6	13	160.0000	0.0000	0.0000	0.0000	0.0000	15.0000	SPAN	SPAN
2	6	11	178.8667	50.0000	0.0000	0.0000	0.0000	13.7500	SPAN	SPAN
3	6	10	197.7333	100.0000	0.0000	0.0000	0.0000	12.5000	SPAN	SPAN
4	6	8	216.6000	150.0000	0.0000	0.0000	0.0000	11.2500	SPAN	SPAN
5	6	6	235.4667	200.0000	0.0000	0.0000	0.0000	10.0000	SPAN	SPAN
6	6	4	254.3333	250.0000	0.0000	0.0000	0.0000	8.7500	SPAN	SPAN
7	6	2	273.2000	300.0000	0.0000	0.0000	0.0000	7.5000	SPAN	SPAN
11	7	16	0.0000	0.0000	0.0000	0.0000	0.0000	15.0000	SPAN	SPAN
12	7	15	38.8667	50.0000	0.0000	0.0000	0.0000	13.7500	SPAN	SPAN
13	7	12	77.7333	100.0000	0.0000	0.0000	0.0000	12.5000	SPAN	SPAN
14	7	9	116.6000	150.0000	0.0000	0.0000	0.0000	11.2500	SPAN	SPAN
15	7	7	155.4667	200.0000	0.0000	0.0000	0.0000	10.0000	SPAN	SPAN
16	7	5	194.3333	250.0000	0.0000	0.0000	0.0000	8.7500	SPAN	SPAN
17	7	3	233.2000	300.0000	0.0000	0.0000	0.0000	7.5000	SPAN	SPAN
20	3	1	579.9890	866.0318	0.0000	0.0000	0.0000	0.0000	SPAN	SPAN
101	8	14	160.0000	0.0000	-45.0000	0.0000	0.0000	0.0000	SPAN	SPAN
111	9	17	0.0000	0.0000	-45.0000	0.0000	0.0000	0.0000	SPAN	SPAN

RUN TIME FOR ABOVE OVERLAY .132 CP SECONDS

- NOTES:
1. Output from PRINT INPUT (NODAL) statement. Line 4 of Control Program (sec. 8.1.2).
 2. The input nodal data are shown by lines 1-11 in section 8.1.3.
 3. Default printout order is by increasing user-assigned node numbers.
 4. All coordinates are printed relative to the GLOBAL reference frame.
 5. A blank in the "reference frame" columns denotes the GLOBAL reference frame.

*** STIFFNESS ELEMENT DATA *** DATA SET NUMBER = 1
 PROBLEM 10 -- EXAMPLE STRESS ANALYSIS

USER ID

7	COVER	INPUT RECORD USER NODES	19, INTERNAL NO. INTERNAL NODES	3,	TEMPERATURE PROPERTIES	70,	MATERIAL CODE	1
		6	4		T(0)U =	.25000		
		7	2		T(1)U =	0.00000		
		17	3		T(2)U =	0.00000		
					ALPHAU =	0.00000		
					BETAU =	0.00000		
					T(0)L =	.25000		
					T(1)L =	0.00000		
					T(2)L =	0.00000		
					ALPHAL =	0.00000		
					BETAL =	0.00000		

10	SPAR	INPUT RECORD USER NODES	14, INTERNAL NO. INTERNAL NODES	18,	TEMPERATURE PROPERTIES	70,	MATERIAL CODE	1
		1	13		T-WEB =	.25000		
		2	11		FAREA1U =	1.50000		
					FAREA1L =	1.50000		
					FAREA2U =	1.50000		
					FAREA2L =	1.50000		
					O(1)U =	0.00000		
					O(1)L =	0.00000		
					O(2)U =	0.00000		
					O(2)L =	0.00000		
					A-LMP1U =	1.25000		
					A-LMP1L =	1.25000		
					A-LMP2U =	1.14583		
					A-LMP2L =	1.14583		

11	SPAR	INPUT RECORD USER NODES	14, INTERNAL NO. INTERNAL NODES	15,	TEMPERATURE PROPERTIES	70,	MATERIAL CODE	1
		2	11		T-WEB =	.25000		
		3	10		FAREA1U =	1.50000		
					FAREA1L =	1.50000		
					FAREA2U =	1.50000		
					FAREA2L =	1.50000		
					O(1)U =	0.00000		
					O(1)L =	0.00000		
					O(2)U =	0.00000		
					O(2)L =	0.00000		
					A-LMP1U =	1.14583		
					A-LMP1L =	1.14583		
					A-LMP2U =	1.04167		
					A-LMP2L =	1.04167		

NOTES:

1. Output from PRINT INPUT (STIFFNESS) statement. Line 5 of Control Program (sec. 8.1.2).
2. The input stiffness data are shown by lines 12-23 in section 8.1.3.
3. Default printout order is by increasing user-assigned element numbers.

DATE - 77/04/20.

PROBLEM ID - EXAMPLE STRESS ANALYSIS

REF. FRAME
OR
SURFACE

DIRECTION VECTOR
OR
ORIENTATION NODE

1	COVER LOADCASE P(P1U) AIRLOAD 3.5000000	1 P(P2U)	12 P(P3U)	11 P(P4U)	P(P1L)	P(P2L)	P(P3L)	P(P4L)	GLOBAL	0.000000 0.000000 1.000000
2	COVER LOADCASE P(P1U) AIRLOAD 3.0000000	1 P(P2U)	2 P(P3U)	13 P(P4U)	P(P1L)	P(P2L)	P(P3L)	P(P4L)	GLOBAL	0.000000 0.000000 1.000000
3	COVER LOADCASE P(P1U) AIRLOAD 2.5000000	2 P(P2U)	3 P(P3U)	14 P(P4U)	P(P1L)	P(P2L)	P(P3L)	P(P4L)	GLOBAL	0.000000 0.000000 1.000000
4	COVER LOADCASE P(P1U) AIRLOAD 2.0000000	3 P(P2U)	4 P(P3U)	15 P(P4U)	P(P1L)	P(P2L)	P(P3L)	P(P4L)	GLOBAL	0.000000 0.000000 1.000000
5	COVER LOADCASE P(P1U) AIRLOAD 1.5000000	4 P(P2U)	5 P(P3U)	16 P(P4U)	P(P1L)	P(P2L)	P(P3L)	P(P4L)	GLOBAL	0.000000 0.000000 1.000000
6	COVER LOADCASE P(P1U) AIRLOAD 1.0000000	5 P(P2U)	6 P(P3U)	17 P(P4U)	P(P1L)	P(P2L)	P(P3L)	P(P4L)	GLOBAL	0.000000 0.000000 1.000000
7	COVER LOADCASE P(P1U) AIRLOAD .5000000	6 P(P2U)	7 P(P3U)	17 P(P4U)	P(P1L)	P(P2L)	P(P3L)	P(P4L)	GLOBAL	0.000000 0.000000 1.000000

NOTES:

1. Output from PRINT INPUT (LOADS) statement. Line 6 of Control Program (sec. 8.1.2).
2. The input element loads data are shown by lines 35-40 in section 8.1.3.
3. Printout of the input element-distributed loads is in an increasing element-number order.
4. Only the pressure intensities that are specified for the points corresponding to the element types are printed.
5. Multiple loadcases with distributed loads would all be shown for each element.

DATE - 77/04/20.

```

***** ELEMENT THERMAL LOADS *****
PROBLEM ID -- EXAMPLE STRESS ANALYSIS
SET NO. 1 STAGE NO. 1

```

1	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	1	12	11	P1L P2L	-200.0000 -200.0000	-200.0000 -200.0000	P3L P4L
2	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	1	2	13	12	P1L P2L	-200.0000 -200.0000	-200.0000 -200.0000
3	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	2	3	14	13	P1L P2L	-200.0000 -200.0000	-200.0000 -200.0000
4	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	3	4	15	14	P1L P2L	-200.0000 -200.0000	-200.0000 -200.0000
5	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	4	5	16	15	P1L P2L	-200.0000 -200.0000	-200.0000 -200.0000
6	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	5	6	17	16	P1L P2L	-200.0000 -200.0000	-200.0000 -200.0000
7	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	6	7	17	17	P1L P2L	-200.0000 -200.0000	-200.0000 -200.0000
10	SPAR LOADCASE THERMAL	MATL 1 N1(U)	ELEM TEMP N1(L) N2(U)	70 N2(L)	NODES N2(L)	1	2			WEB		
11	SPAR LOADCASE THERMAL	MATL 1 N1(U)	ELEM TEMP N1(L) N2(U)	70 N2(L)	NODES N2(L)	2	3			WEB		
12	SPAR LOADCASE THERMAL	MATL 1 N1(U)	ELEM TEMP N1(L) N2(U)	70 N2(L)	NODES N2(L)	3	4			WEB		
13	SPAR LOADCASE THERMAL	MATL 1 N1(U)	ELEM TEMP N1(L) N2(U)	70 N2(L)	NODES N2(L)	4	5			WEB		
14	SPAR LOADCASE THERMAL	MATL 1 N1(U)	ELEM TEMP N1(L) N2(U)	70 N2(L)	NODES N2(L)	5	6			WEB		

NOTES:

1. Output from PRINT INPUT (LOADS) statement. Line 6 of Control Program (sec. 8.1.2).
2. The input element thermal loads data are shown by lines 41-45 in section 8.1.3.
3. Similar format as shown on the preceding page for the input element distributed loads. In this printout, however, the nodal delta-temperatures are shown.

*** B O U N D A R Y C O N D I T I O N D A T A ***
 DATA SET NUMBER 1, EXECUTION STAGE 1 DATE 77/04/20 PAGE 1
 PROBLEM ID -- EXAMPLE STRESS ANALYSIS

COORDINATE SYSTEM REFERENCE TABLE

SYSTEM (USERID)	TYPE	FREEDOM LABELS						FREEDOM-FORCE LABELS					
		1	2	3	4	5	6	1	2	3	4	5	6
1. GLOBAL	REC												
2. SPAN	REC												

THE CODE IN THE FOLLOWING TABLE IS AS FOLLOWS

CHARACTER F DENOTES A PARTITION 1 FREEDOM (F = FREE)
 CHARACTER R DENOTES A PARTITION 2 FREEDOM (R = RETAIN)
 CHARACTER S DENOTES A PARTITION 3 FREEDOM (S = SUPPORT)
 A BLANK DENOTES A FREEDOM WITH ZERO STIFFNESS

USER NODE	INTERNAL NODE	INPUT ORDER	FREEDOM CODE						ANALYSIS REF. FRAME
			1	2	3	4	5	6	
1	13	2	S	S	S	S	S	S	SPAN
2	11	3	F	F	F	F	F	F	SPAN
3	10	4	F	F	F	F	F	F	SPAN
4	8	5	F	F	F	F	F	F	SPAN
5	6	6	F	F	F	F	F	F	SPAN
6	4	7	F	F	F	F	F	F	SPAN
7	2	8	F	F	F	F	F	F	SPAN
11	16	9	S	S	S	S	S	S	SPAN
12	15	10	F	F	F	F	F	F	SPAN
13	12	11	F	F	F	F	F	F	SPAN
14	9	12	F	F	F	F	F	F	SPAN
15	7	13	F	F	F	F	F	F	SPAN
16	5	14	F	F	F	F	F	F	SPAN
17	3	15	F	F	F	F	F	F	SPAN
20	1	1	S	S	S	S	S	S	SPAN
101	14	16	S	S	S	S	S	S	SPAN
111	17	17	S	S	S	S	S	S	SPAN

THE STIFFNESS MODULE HAS BEEN EXECUTED FOR THIS DATA SET AND STAGE.

NOTES:

1. Output from PRINT INPUT (BC) statement. Line 8 of Control Program-(sec. 8.1.2).
2. The input BC data are shown by lines 24-26 in section 8.1.3.
3. Default printout order is by increasing user-assigned node numbers.
4. Inactive freedoms are identified by blanks in the "freedom-code" columns. For this to be shown in the BC printout, execution of the STIFFNESS Processor must have been done previously (e.g. by the PERFORM STRESS statement).

***** RESULTANT ABOUT THE GLOBAL ORIGIN OF LOADS APPLIED TO SET 1 STAGE 1 *****

LOADCASE	F1	F2	F3	M1	M2	M3
AIRLOAD	0.	0.	.69522E+05	.63770E+07	-.91050E+07	0.
TIPLoad	0.	0.	-.20000E+04	-.60000E+06	.50640E+06	0.
THERMAL	-.16880E-08	.37835E-08	-.58935E-09	-.55879E-08	.59605E-07	.27861E-05

NOTES:

1. Output from PRINT OUTPUT (LOADS) statement. Line 9 of Control Program (sec. 8.1.2).

a) Resultants of the applied loads for 6 degrees of freedom and for each loadcase, as shown on this page.

b) The cumulated equivalent nodal loads as applied to the 6 degrees of freedom of each node for each loadcase. These data are printed in three blocks corresponding to the FREE, RETAIN, and SUPPORT nodal freedoms. An example is shown on the next page.

LOADS (FREE PARTITION)-PROB ID -EXAMPLE STRESS ANALYSIS

DATA SET NO. 1, STAGE NO. 1

INTERNAL NODE NO.	USER NODE NO.	ANALYSIS FRAME	LOADCASE	1	2	3	4	5	6
2	7	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 29006.86	0.000000 0.000000 -20105.2	166.7187 -1000.00 -6727.27	0.000000 0.000000 2628747.	0.000000 0.000000 3787959.	0.000000 0.000000 0.000000
3	17	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 24630.34	0.000000 0.000000 46194.63	916.9531 -1000.00 -19469.2	0.000000 0.000000 -60E+07	0.000000 0.000000 3217306.	0.000000 0.000000 0.000000
4	6	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 9024.575	0.000000 0.000000 -54843.9	833.5937 0.000000 8502.081	0.000000 0.000000 8366123.	0.000000 0.000000 1366329.	0.000000 0.000000 0.000000
5	16	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 8974.607	0.000000 0.000000 65235.43	2334.062 0.000000 -17632.3	0.000000 0.000000 -1.0E+08	0.000000 0.000000 1361969.	0.000000 0.000000 0.000000
6	5	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 9024.575	0.000000 0.000000 -54843.9	2125.664 0.000000 8499.295	0.000000 0.000000 9561449.	0.000000 0.000000 1560570.	0.000000 0.000000 0.000000
7	15	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 8974.607	0.000000 0.000000 65235.43	4126.289 0.000000 -17634.6	0.000000 0.000000 -1.1E+08	0.000000 0.000000 1555682.	0.000000 0.000000 0.000000
8	4	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 9024.575	0.000000 0.000000 -54843.9	3834.531 0.000000 8496.895	0.000000 0.000000 1.08E+08	0.000000 0.000000 1754675.	0.000000 0.000000 0.000000
9	14	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 8974.607	0.000000 0.000000 65235.43	6418.672 0.000000 -17636.7	0.000000 0.000000 -1.13E+08	0.000000 0.000000 1749254.	0.000000 0.000000 0.000000
10	3	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 9024.575	0.000000 0.000000 -54843.9	6043.554 0.000000 8494.811	0.000000 0.000000 1.12E+08	0.000000 0.000000 1948663.	0.000000 0.000000 0.000000
11	2	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 9024.575	0.000000 0.000000 -54843.9	8752.734 0.000000 8492.991	0.000000 0.000000 1.13E+08	0.000000 0.000000 2142547.	0.000000 0.000000 0.000000
12	13	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 8974.607	0.000000 0.000000 65235.43	9211.210 0.000000 -17638.6	0.000000 0.000000 -1.14E+08	0.000000 0.000000 1942703.	0.000000 0.000000 0.000000

DISPLACEMENTS *** PROBLEM ID -- EXAMPLE STRESS ANALYSIS

DATA SET NO. 1

INTERNAL NODE NO.	USER NODE NO.	ANALYSIS FRAME	LOADCASE	TX	TY	TZ	PX	PY	PZ
13	1	SPAN	AIRLOAD TIPLoad THERMAL	0.00000 0.00000 0.00000	0.00000 0.00000 0.00000	0.00000 0.00000 0.00000	0.00000 0.00000 0.00000	0.00000 0.00000 0.00000	0.00000 0.00000 0.00000
11	2	SPAN	AIRLOAD TIPLoad THERMAL	-0.00369 0.000436 0.0106724	0.002981 -8.2E-05 -0.006688	0.1161881 -0.008259 -0.269957	-0.01220 0.000887 0.071189	-0.01192 0.007699 0.122999	0.00000 0.00000 0.00000
10	3	SPAN	AIRLOAD TIPLoad THERMAL	-0.000994 0.001044 0.0207017	0.004041 8.30E-05 -0.008005	0.2990050 -0.01371 -1.13446	-0.02204 0.001687 0.086926	-0.03726 0.005760 0.241326	-0.000946 0.000853 0.0039541
8	4	SPAN	AIRLOAD TIPLoad THERMAL	-0.000987 0.000997 0.0284040	0.003603 0.000369 -0.008733	0.5466834 -0.072004 -2.66783	-0.002594 0.002148 0.0089727	-0.05172 0.009115 0.0372266	0.000000 0.000000 0.000000
6	5	SPAN	AIRLOAD TIPLoad THERMAL	-0.000969 0.000939 0.0360684	0.002513 0.000724 -0.009222	0.8480224 -0.131534 -4.92164	-0.001963 0.001878 0.072949	-0.06211 0.012545 0.0512769	0.000000 0.000000 0.000000
4	6	SPAN	AIRLOAD TIPLoad THERMAL	-0.000952 0.000881 0.0437348	0.001473 0.001074 -0.009729	1.192507 -0.210655 -7.96195	0.002410 0.967E-05 0.032380	-0.006643 0.015533 0.0661657	0.000000 0.000000 0.000000
2	7	SPAN	AIRLOAD TIPLoad THERMAL	-0.000935 0.000824 0.0514015	0.000427 0.001424 -0.10233	1.562481 -0.305187 -11.8560	0.043454 -0.00401 -0.03235	-0.006178 0.016165 0.0818392	0.000000 0.000000 0.000000
16	11	SPAN	AIRLOAD TIPLoad THERMAL	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000
15	12	SPAN	AIRLOAD TIPLoad THERMAL	-0.000444 0.000285 0.0073171	0.002412 -0.000016 0.0077311	0.2113850 -0.008925 -0.156644	-0.001532 0.001287 -0.010691	-0.02084 0.002031 0.0071389	0.000000 0.000000 0.000000
12	13	SPAN	AIRLOAD TIPLoad THERMAL	-0.000994 0.000575 0.0164378	0.005513 -0.000030 0.0083522	0.5094068 -0.032429 -0.776470	-0.002831 0.002792 -0.13541	-0.03280 0.004078 0.169653	-0.000655 0.000616 -0.004286
9	14	SPAN	AIRLOAD TIPLoad THERMAL	-0.000805 0.000411 0.0241875	0.005094 -0.16E-05 0.0048371	0.8393019 -0.071953 -2.03769	-0.03751 0.003957 -0.10064	-0.03713 0.006333 0.0274061	0.000000 0.000000 0.000000

NOTES:

1. Output from the PRINT OUTPUT (DISP) statement. Line 10 of Control Program (sec. 8.1.2).
2. Nodal displacements are shown for each loadcase.

STRESSES ** PROBLEM ID -- EXAMPLE STRESS ANALYSIS

DATA SET NO. 1

COVER NODES= 5 6 17 16 MATL=M 1 TEMP= 70
T(0)U=.250000 T(1)U=0.00000 T(2)U=0.00000 ALPHAU=0.00000 BETAU=0.00000 T(0)L=.250000
T(1)L=0.00000 T(2)L=0.00000 ALPHAL=0.00000 BETAL=0.00000

LOADCASE SIGMA1U SIGMA2U SIG-S1U SIG-S2U SIGMAIL SIGMA2L TAU12L SIG-S1L SIG-S2L
AIRLOAD -427.677 136.4094 -169.611 0.00000 0.00000 427.7017 -136.346 169.6498 0.00000 0.00000
TIPLDAD 376.9950 -31.2927 -40.1406 0.00000 0.00000 -376.998 31.28601 40.13651 0.00000 0.00000
THERMAL 462.1270 -216.817 445.4912 0.00000 0.00000 -461.703 216.5958 -445.693 0.00000 0.00000

COVER NODES= 6 7 17 MATL=M 1 TEMP= 70
T(0)U=.250000 T(1)U=0.00000 T(2)U=0.00000 ALPHAU=0.00000 BETAU=0.00000 T(0)L=.250000
T(1)L=0.00000 T(2)L=0.00000 ALPHAL=0.00000 BETAL=0.00000

LOADCASE SIGMA1U SIGMA2U SIG-S1U SIG-S2U SIGMAIL SIGMA2L TAU12L SIG-S1L SIG-S2L
AIRLOAD -330.814 166.9083 -82.1131 0.00000 0.00000 330.8036 -166.920 82.10167 0.00000 0.00000
TIPLDAD 232.6912 -44.4240 -36.1073 0.00000 0.00000 -232.690 44.42524 36.10853 0.00000 0.00000
THERMAL 583.0639 -334.324 165.8385 0.00000 0.00000 -582.328 334.2965 -165.892 0.00000 0.00000

SPAR NODES= 1 2 MATL=M 1 TEMP= 70
T-WEB=.250000 FAREAU=1.50000 FAREAL=1.50000 FAREAU=1.50000 FAREAL=1.50000 T(0)U=0.00000
O(1)U=0.00000 O(2)U=0.00000 C(2)L=0.00000 A-LMP1U=1.25000 A-LMP2U=1.14583
A-LMP2L=1.14583

LOADCASE P-CAPU SIGMA-U P-LMPU P-CAPL SIGMA-L P-LMPL Q-EQUIV TAU-MAX
AIRLOAD -7976.27 -2834.32 -3724.79 7791.734 2765.892 3642.895 644.1460 2810.819
TIPLDAD 1079.304 397.0423 483.7404 -1055.00 -388.032 -472.956 -16.5046 -72.0201
THERMAL 7609.198 -9080.30 21229.64 -6466.94 8210.764 -18783.1 65.82426 287.2332

SPAR NODES= 2 3 MATL=M 1 TEMP= 70
T-WEB=.250000 FAREAU=1.50000 FAREAL=1.50000 FAREAU=1.50000 FAREAL=1.50000 T(0)U=0.00000
O(1)U=0.00000 O(2)U=0.00000 C(2)L=0.00000 A-LMP1U=1.14583 A-LMP2U=1.04167
A-LMP2L=1.04167

LOADCASE P-CAPU SIGMA-U P-LMPU P-CAPL SIGMA-L P-LMPL Q-EQUIV TAU-MAX
AIRLOAD -6435.83 -2394.75 -2843.70 6080.074 2259.849 2696.300 422.8658 1860.609
TIPLDAD 1138.463 435.0027 485.9590 -1101.87 -420.887 -470.535 -19.8055 -87.1443
THERMAL 6903.016 -8635.60 19856.42 -5623.18 7901.289 -17475.1 78.98912 347.5521

SPAR NODES= 3 4 MATL=M 1 TEMP= 70
T-WEB=.250000 FAREAU=1.50000 FAREAL=1.50000 FAREAU=1.50000 FAREAL=1.50000 T(0)U=0.00000
O(1)U=0.00000 O(2)U=0.00000 C(2)L=0.00000 A-LMP1U=1.04167 A-LMP2U=.937500
A-LMP2L=.937500

LOADCASE P-CAPU SIGMA-U P-LMPU P-CAPL SIGMA-L P-LMPL Q-EQUIV TAU-MAX
AIRLOAD -4957.57 -1926.41 -2067.96 4957.615 1926.429 2067.972 292.7769 1301.231
TIPLDAD 1141.855 452.3655 463.3067 -1141.86 -452.367 -463.308 -28.8760 -128.338
THERMAL 4529.482 -8055.58 18612.56 -6533.79 6896.298 -16878.2 -66.8167 -296.963

NOTES:

1. Output from the PRINT OUTPUT (STRESS) statement. Line 11 of Control Program (sec. 8.1.2).
2. Element stresses are shown for each loadcase.

8.2 EXAMPLE 2: BIFURCATION BUCKLING

8.2.1 Description of Model

This example demonstrates the use of ATLAS to perform a bifurcation buckling analysis of the plane frame shown in figure 8-4. The columns have a cross-sectional area of 5 and a bending moment of inertia of 10. The beam has a cross-sectional area of 100 and a bending moment of inertia of 20.

8.2.2 Description of Analysis

The loading consists of concentrated forces at the column tops as shown in figure 8-4. All unsupported active degrees of freedom (TX, TY and RZ) are retained in this analysis.

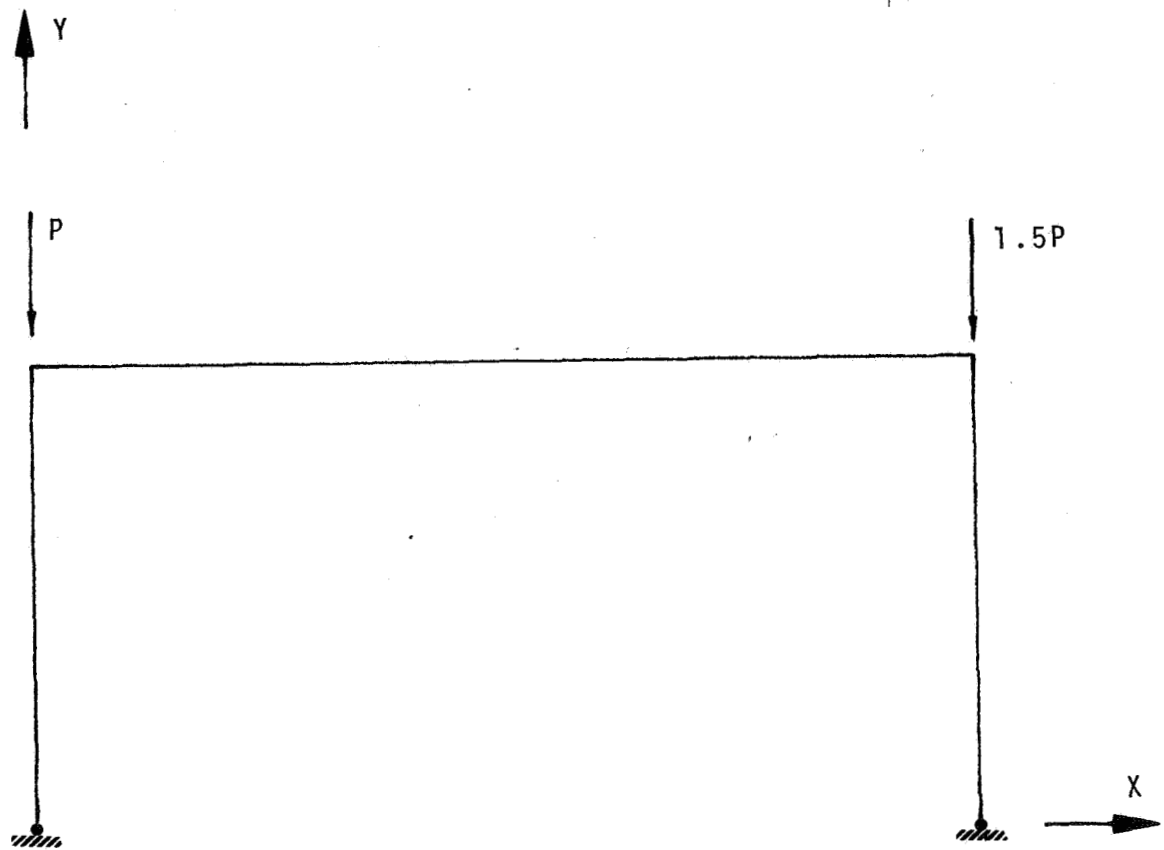


Figure 8-4. Plane Frame Model

8.2.3 Execution-Control Data

```
1 BEGIN CONTROL PROGRAM BUCKLE
2 PROBLEM ID (EXAMPLE BUCKLING PROBLEM)
3 READ INPUT (MODE2)
4 PRINT INPUT (NODAL)
5 PRINT INPUT (STIFFNESS)
6 PERFORM R-STRESS
7 PRINT INPUT (BC)
8 PRINT OUTPUT (STRESS)
9 EXECUTE STIFFNESS (LC=COLMLD)
10 EXECUTE MERGE (GSTIF,KG22=22)
11 EXECUTE BUCKLING (STIF=KRED,KG=KG22)
12 PRINT OUTPUT (BUCKLING)
13 END CONTROL PROGRAM
```

NOTES:

- ① Required first statement in an ATLAS execution-control deck. The name selected for this Control Program is BUCKLE.
- ② Assigns a title to the problem. This title will appear in the printout.
- ③ Directs ATLAS to read and check the model-definition data deck.
- ④⑤ Request formatted printout of the input data for:
 - ④ nodes
 - ⑤ stiffness elements
- ⑥ Directs ATLAS to perform a stress analysis. The R-STRESS execution control procedure calculates the elastic stiffnesses, forms the equilibrium equations, and solves for displacements, stresses, and reactions.
- ⑦⑧ Request formatted printout of:
 - ⑦ boundary condition data
 - ⑧ element stresses
- ⑨ Directs ATLAS to execute the STIFFNESS processor for calculation of the element geometric stiffness matrices for loadcase COLMLD
- ⑩ Directs ATLAS to execute the MERGE processor for creation of the geometric stiffness matrix KG22

- ⑪ Directs ATLAS to execute the BUCKLING processor for calculation of the buckling loads and mode shapes.
- ⑫ Requests a formatted printout of the buckling eigenvalues (critical loadcase factors) and mode shapes
- ⑬ Required last statement in an ATLAS execution-control deck

8.2.4 Model-Definition Data

```

1 BEGIN NODAL DATA
2     1  0.  0.  0.  TO  3  0.  5.  0.
3     7 10.  0.  0.  TO  5 10.  5.  0.
4     REORDER FROM -1
5     3                      TO  5
6 END NODAL DATA

7 BEGIN STIFFNESS DATA
8     BEGIN ELEMENT DATA
9         BEAM N1  1  2  5      5.  10.  TO  N2  2  3  5
10        BEAM N3  3  4  7    100.  20.  TO  N4  4  5  7
11        BEAM N5  5  6  1      5.  10.  TO  N6  6  7  1
12        END ELEMENT DATA
13    END STIFFNESS DATA

14 BEGIN BC DATA
15     SUPPORT TX TY FOR 1 7
16     RETAIN RZ FOR 1 7
17     RETAIN TX TY RZ FOR 2 TO 6
18 END BC DATA

19 BEGIN LOADS DATA
20     LOAD CASE ID COLMLD *# RT. COL. LOAD = 1.5 LD. ON LFT. COL.*
21     BEGIN NODAL LOAD DATA
22         CASE COLMLD
23             ORDER FY
24             3  -1.
25             5  -1.5
26     END NODAL LOAD DATA
27 END LOADS DATA

28 END PROBLEM DATA

```

NOTES:

- ① Identifies the start of the NODAL data set
- ②,③ Define nodes 1,2,3 and 5,6,7 for column members

- ④ Specifies that nodal data are not to be sorted prior to problem solution
- ⑤ Generates node 4 for beam member
- ⑥ Identifies the end of the NODAL data set
- ⑦ Identifies the start of the STIFFNESS data set
- ⑧ Identifies the start of the structural element data subset
- ⑨-⑪ Define six BEAM elements; columns are element numbers 1,2,5 and 6; the horizontal beam is element numbers 3 and 4; only area and IZ section properties are specified for the planar structure; all inactive freedoms are ignored during problem solution.
- ⑫ Identifies the end of the structural element data
- ⑬ Identifies the end of the STIFFNESS data set
- ⑭ Identifies the start of the boundary condition (BC) data set
- ⑮ Defines translational freedoms in the X and Y directions at nodes 1 and 7 to be supported (fixed)
- ⑯,⑰ Define all remaining active degrees of freedom (TX,TY and RZ) to be retained in the reduced structural matrices for the buckling analysis
- ⑱ Identifies the end of the BC data
- ⑲ Identifies the start of the LOADS data set
- ⑳ Defines a title for the loadcase COLMLD
- ㉑ Identifies the start of a NODAL-LOAD data subset
- ㉒ Identifies COLMLD as the loadcase for subsequent loads
- ㉓ Specifies that only forces in the GLOBAL Y-direction are being input
- ㉔,㉕ Define loads of -1.0 and -1.5 at nodes 3 and 5, respectively
- ㉖ Identifies the end of the NODAL LOAD data

- ②⑦ Identifies the end of the LOADS data set
- ②⑧ Identifies the end of the model-definition data deck

8.2.5 Results

Representative output from the buckling analysis is shown on the following pages. Printout of input data (nodal, stiffness, and boundary condition) and output data (stresses, buckling eigenvalues, and mode shapes) are included.

***** NODAL DATA ***** DATA SET NUMBER = 1
 PROBLEM ID --- EXAMPLE BUCKLING PROBLEM

USER NODE	INPUT RECORD	INTERNAL NODE	X COORD	Y COORD	Z COORD	DX COORD	DY COORD	DZ COORD
1	2	1	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
2	2	2	0.0000	2.5000	0.0000	0.0000	0.0000	0.0000
3	2	3	0.0000	5.0000	0.0000	0.0000	0.0000	0.0000
4	5	4	5.0000	5.0000	0.0000	0.0000	0.0000	0.0000
5	3	5	10.0000	5.0000	0.0000	0.0000	0.0000	0.0000
6	3	6	10.0000	2.5000	0.0000	0.0000	0.0000	0.0000
7	3	7	10.0000	0.0000	0.0000	0.0000	0.0000	0.0000

RUN TIME FOR ABOVE OVERLAY .075 CP SECONDS

NOTES:

1. Output from PRINT INPUT (NODAL) statement. Line 4 of Control Program (sec. 8.2.3).
2. The input nodal data are shown by lines 1-6 in section 8.2.4.
3. Default printout order is by increasing user-assigned node numbers.
4. All coordinates are printed relative to the GLOBAL reference frame.

*** STIFFNESS ELEMENT DATA *** DATA SET NUMBER = 1
 PROBLEM ID -- EXAMPLE BUCKLING PROBLEM

USER ID

1	BEAM	INPUT RECORD USER NODES	9, INTERNAL NO. INTERNAL NODES	1, TEMPERATURE PROPERTIES	70, MATERIAL CODE
1	1	1	1	A(1) = 5.00000	1
		2	2	A-VY(1) = 0.00000	
		5	5	A-VZ(1) = 0.00000	
				J(1) = 0.00000	
				IY(1) = 0.00000	
				IZ(1) = 10.00000	
				A(2) = 5.00000	
				A-VY(2) = 0.00000	
				A-VZ(2) = 0.00000	
				J(2) = 0.00000	
				IY(2) = 0.00000	
				IZ(2) = 10.00000	
				CONSTR =	0.

2	BEAM	INPUT RECORD USER NODES	9, INTERNAL NO. INTERNAL NODES	2, TEMPERATURE PROPERTIES	70, MATERIAL CODE
2	2	2	2	A(1) = 5.00000	1
		3	3	A-VY(1) = 0.00000	
		5	5	A-VZ(1) = 0.00000	
				J(1) = 0.00000	
				IY(1) = 0.00000	
				IZ(1) = 10.00000	
				A(2) = 5.00000	
				A-VY(2) = 0.00000	
				A-VZ(2) = 0.00000	
				J(2) = 0.00000	
				IY(2) = 0.00000	
				IZ(2) = 10.00000	
				CONSTR =	0.

3	BEAM	INPUT RECORD USER NODES	10, INTERNAL NO. INTERNAL NODES	3, TEMPERATURE PROPERTIES	70, MATERIAL CODE
3	3	3	3	A(1) = 100.00000	1
		4	4	A-VY(1) = 0.00000	
		7	7	A-VZ(1) = 0.00000	
				J(1) = 0.00000	
				IY(1) = 0.00000	
				IZ(1) = 20.00000	
				A(2) = 100.00000	
				A-VY(2) = 0.00000	
				A-VZ(2) = 0.00000	
				J(2) = 0.00000	
				IY(2) = 0.00000	
				IZ(2) = 20.00000	
				CONSTR =	0.

NOTES:

1. Output from PRINT INPUT (STIFFNESS) statement. Line 5 of Control Program (sec. 8.2.3).
2. The input stiffness data are shown by lines 7-13 in section 8.2.4.
3. Default printout order is by increasing user-assigned element numbers.

*** B O U N D A R Y C O N D I T I O N D A T A ***

DATA SET NUMBER 1, EXECUTION STAGE 1 DATE 77/04/15 PAGE 1
 PROBLEM ID -- EXAMPLE BUCKLING PROBLEM

THE CODE IN THE FOLLOWING TABLE IS AS FOLLOWS

CHARACTER F DENOTES A PARTITION 1 FREEDOM (F = FREE)
 CHARACTER R DENOTES A PARTITION 2 FREEDOM (R = RETAIN)
 CHARACTER S DENOTES A PARTITION 3 FREEDOM (S = SUPPORT)
 A BLANK DENOTES A FREEDOM WITH ZERO STIFFNESS

USER NODE	INTERNAL NODE	INPUT ORDER	FREEDOM CODE					
			1	2	3	4	5	6
1	1	1	S	S				R
2	2	2	R	R				R
3	3	3	R	R				R
4	4	7	R	R				R
5	5	6	R	R				R
6	6	5	R	R				R
7	7	4	S	S				R

THE STIFFNESS MODULE HAS BEEN EXECUTED FOR THIS DATA SET AND STAGE.

NOTES:

1. Output from PRINT INPUT (BC) statement. Line 7 of Control Program (sec. 8.2.3).
2. The input BC data are shown by lines 14-18 in section 8.2.4.
3. Default printout order is by increasing user-assigned node numbers.
4. Inactive freedoms are identified by blanks in the "freedom-code" columns. For this to be shown in the BC printout, execution of the STIFFNESS Processor must have been done previously (e.g. by the PERFORM R-STRESS statement in line 6 of sec. 8.2.3).
5. A list of retained freedoms is also printed separately as shown on the next page.

*** RETAINED FREEDOM DATA ***

DATA SET NUMBER 1, EXECUTION STAGE 1
 PROBLEM ID -- EXAMPLE BUCKLING PROBLEM

DATE 77/04/15 PAGE 2

CUMULATIVE COUNT	INTERNAL NODE	USER NODE	INPUT ORDER	FREEDOM CODE
1	1	1	1	RZ
2	7	7	4	RZ
3	2	2	2	TX
4	2	2	2	TY
5	2	2	2	RZ
6	3	3	3	TX
7	3	3	3	TY
8	3	3	3	RZ
9	4	4	7	TX
10	4	4	7	TY
11	4	4	7	RZ
12	5	5	6	TX
13	5	5	6	TY
14	5	5	6	RZ
15	6	6	5	TX
16	6	6	5	TY
17	6	6	5	RZ

RUN TIME FOR ABOVE OVERLAY .109 CP SECONDS

NOTES:

1. Output from PRINT INPUT (BC) statement. Line 7 of Control Program (sec. 8.2.3).
2. The retained degrees of freedom as defined by the input BC data (lines 14-18 in sec. 8.2.4) are listed in the user-specified order.

STRESSES ** PROBLEM ID --- EXAMPLE BUCKLING PROBLEM

DATA SET NO. 1

BEAM 1
NODES= 1 2 5 MATL=M 1 TEMP= 70
A11=5.00000 A-VY(1)=0.00000 A-VZ(1)=0.00000 J(1)=0.00000 IV(1)=0.00000 IZ(1)=10.0000
A12=5.00000 A-VY(2)=0.00000 A-VZ(2)=0.00000 J(2)=0.00000 IV(2)=0.00000 IZ(2)=10.0000
CONSTR=0.00000

LOADCASE COLMID P(2) VY(2) VZ(2) T(2) MY(1) MZ(1) MY(2) MZ(2)
-1.00000 .409E-13 0.00000 0.00000 0.00000 0.00000 -.18E-14 -.82E-13

BEAM 2
NODES= 2 3 5 MATL=M 1 TEMP= 70
A11=5.00000 A-VY(1)=0.00000 A-VZ(1)=0.00000 J(1)=0.00000 IV(1)=0.00000 IZ(1)=10.0000
A12=5.00000 A-VY(2)=0.00000 A-VZ(2)=0.00000 J(2)=0.00000 IV(2)=0.00000 IZ(2)=10.0000
CONSTR=0.00000

LOADCASE COLMID P(2) VY(2) VZ(2) T(2) MY(1) MZ(1) MY(2) MZ(2)
-1.00000 .568E-13 0.00000 0.00000 0.00000 0.00000 .622E-13 -.19E-12

BEAM 3
NODES= 3 4 7 MATL=M 1 TEMP= 70
A11=100.000 A-VY(1)=0.00000 A-VZ(1)=0.00000 J(1)=0.00000 IV(1)=0.00000 IZ(1)=20.0000
A12=100.000 A-VY(2)=0.00000 A-VZ(2)=0.00000 J(2)=0.00000 IV(2)=0.00000 IZ(2)=20.0000
CONSTR=0.00000

LOADCASE COLMID P(2) VY(2) VZ(2) T(2) MY(1) MZ(1) MY(2) MZ(2)
0.00000 -.24E-13 0.00000 0.00000 0.00000 0.00000 .172E-12 -.11E-13

BEAM 4
NODES= 4 5 7 MATL=M 1 TEMP= 70
A11=100.000 A-VY(1)=0.00000 A-VZ(1)=0.00000 J(1)=0.00000 IV(1)=0.00000 IZ(1)=20.0000
A12=100.000 A-VY(2)=0.00000 A-VZ(2)=0.00000 J(2)=0.00000 IV(2)=0.00000 IZ(2)=20.0000
CONSTR=0.00000

LOADCASE COLMID P(2) VY(2) VZ(2) T(2) MY(1) MZ(1) MY(2) MZ(2)
0.00000 -.28E-13 0.00000 0.00000 0.00000 0.00000 0.00000 .185E-12

BEAM 5
NODES= 5 6 1 MATL=M 1 TEMP= 70
A11=5.00000 A-VY(1)=0.00000 A-VZ(1)=0.00000 J(1)=0.00000 IV(1)=0.00000 IZ(1)=10.0000
A12=5.00000 A-VY(2)=0.00000 A-VZ(2)=0.00000 J(2)=0.00000 IV(2)=0.00000 IZ(2)=10.0000
CONSTR=0.00000

LOADCASE COLMID P(2) VY(2) VZ(2) T(2) MY(1) MZ(1) MY(2) MZ(2)
-1.50000 .409E-13 0.00000 0.00000 0.00000 0.00000 -.17E-12 .817E-13

NOTES:

1. Output from the PRINT OUTPUT (STRESS) statement. Line 8 of Control Program (sec. 8.2.3).

EIGENVALUES WITH INDEX NAME = EIGEN01

MODE	EIGENVALUE
1	.5254760E+07
2	.3337881E+08
3	.3466667E+08
4	.4064876E+08
5	.5067852E+08
6	.5200000E+08
7	.7508608E+08
8	.1592485E+09
9	.2408013E+09
10	.4173009E+09
11	.6209720E+09
12	.7546562E+09
13	.8739776E+09
14	.1131819E+10
15	0.
16	0.
17	0.

NOTES:

1. Output from the PRINT OUTPUT (BUCKLING) statement. Line 12 of Control Program (sec. 8.2.3).
2. The buckling mode shapes corresponding to these buckling eigenvalues are also printed separately as shown on the next page.

SOLUTION MODE SHAPES
WITH INDEX NAME = MODES01

ROW	NODE	DOF	MODE 1	MODE 2	MODE 3	MODE 4	MODE 5	MODE 6	MODE 7	MODE 8
1	1	6	-.26158915	-.03137983	-.00000000	-.16072956	-.2665848	-.00000000	-.36698341	-.21661759
2	7	6	-.26636804	-.22275649	-.00000000	-.26369271	-.05410605	-.00000000	-.15742769	-.84628125
3	2	1	.61326146	.04367801	.00000000	.28074304	-.40643382	-.00000000	-.41833954	-.00559254
4	2	2	.07018310	-.05187074	-.00000000	.27949795	-.50000000	1.00000000	.50000000	-.15516901
5	2	6	-.21324700	.00764632	-.00000000	-.02680090	-.02055823	-.00000000	.14765562	-.24871045
6	3	1	.99733202	-.04917584	.00000000	.13073586	.01490747	-.00000000	-.23513838	.18285933
7	3	2	.14036619	-.10374147	.00000000	.55899589	1.00000000	.00000000	1.00000000	-.31033802
8	3	6	-.08308706	-.05995584	-.00000000	.11074322	-.21550360	-.00000000	-.14200948	.20839763
9	4	1	.99866601	-.04677914	.00000000	.12585636	.01986837	.00000000	-.224416407	.16981304
10	4	2	-.00818509	.28088578	-.00000000	1.00000000	.25146808	.00000000	.54736832	1.00000000
11	4	6	-.00262131	.10263460	-.00000000	.04023657	-.08920354	-.00000000	-.23056985	.18219103
12	5	1	1.00000000	-.04438244	.00000000	.12097686	-.02482927	.00000000	-.25318976	.15675675
13	5	2	-.14872553	1.00000000	-.00000000	.70714300	.05502078	-.00000000	-.38077544	.178.0670
14	5	6	-.07988273	.19275063	-.00000000	-.18280122	.00533024	.00000000	-.04819535	-.64429490
15	6	1	.62138291	.33064173	-.00000000	-.35769500	.07559881	.00000000	.06084166	-.14212691
16	6	2	-.07436276	.50000000	1.00000000	.35357150	.02751039	.00000000	-.19038772	.08905335
17	6	6	-.21376067	.02281271	-.00000000	-.08477224	.00736181	-.00000000	.16089382	.81694123

ROW	NODE	DOF	MODE 9	MODE 10	MODE 11	MODE 12	MODE 13	MODE 14	MODE 15	MODE 16
1	1	6	.94405605	-.21045565	-.78503188	-.52851193	-.11609675	.53889648	-.00000000	-.00000000
2	7	6	.25567217	.79322502	-.25651904	.55458166	.14640901	.02521750	-.00000000	-.00000000
3	2	1	.20510830	-.03787706	-.09779818	-.07774376	.50110595	-.07709860	.00000000	-.00000000
4	2	2	.11765465	-.04931909	.02832724	.05480992	-.00015673	.03300735	.00000000	.00000000
5	2	6	-.93789392	.06735665	-.20877565	-.29508876	-.13291145	.63435759	-.00000000	-.00000000
6	3	1	-.10803025	.09073426	.05176055	-.09260386	1.00000000	-.16619784	.00000000	-.00000000
7	3	2	.23530938	-.09863818	.05665449	.10978185	-.00031345	.06601470	.00000000	.00000000
8	3	6	.69975860	.13459056	.64496076	.19995877	-.17647499	.95571221	.00000000	-.00000000
9	4	1	-.14100193	.07268694	-.02598008	-.03357882	.16686320	-.06968746	-.02466763	1.00000000
10	4	2	1.00000000	1.00000000	1.00000000	-.98087359	-.43621011	1.00000000	1.00000000	.77292214
11	4	6	-.21589074	.16050326	-.13847665	-.32717086	.00108407	-.29652551	-.24100904	.25690371
12	5	1	-.17397361	.05463962	-.10372071	-.07455379	-.66627360	.02682291	.00000000	.00000000
13	5	2	-.14368188	.06277991	-.03665305	-.07142594	.00020465	-.04331451	.00000000	.00000000
14	5	6	-.06359039	-.67975275	-.14703867	1.00000000	.17244958	.16479229	.00000000	-.00000000
15	6	1	.02179546	.15167308	-.05285822	-.03085797	-.33239603	.02005116	.00000000	.00000000
16	6	2	-.07184094	.03138996	-.01832652	-.03571297	.00010233	-.02165726	.00000000	.00000000
17	6	6	-.11605952	.20585510	-.22762477	.65658378	.15187051	-.05108152	-.00000000	.00000000

NOTES:

1. Output from the PRINT OUTPUT (BUCKLING) statement. Line 12 of Control Program (sec. 8.2.3).
2. The row sequence of the buckling mode shapes corresponds to the user-specified sequence of retained freedoms.

8.3 EXAMPLE 3: WEIGHTS ANALYSIS

8.3.1 Description of Model

This example demonstrates the use of ATLAS to calculate the weight of the panels shown in figure 8-5 and print a weight statement for the wing box described in section 8.0.

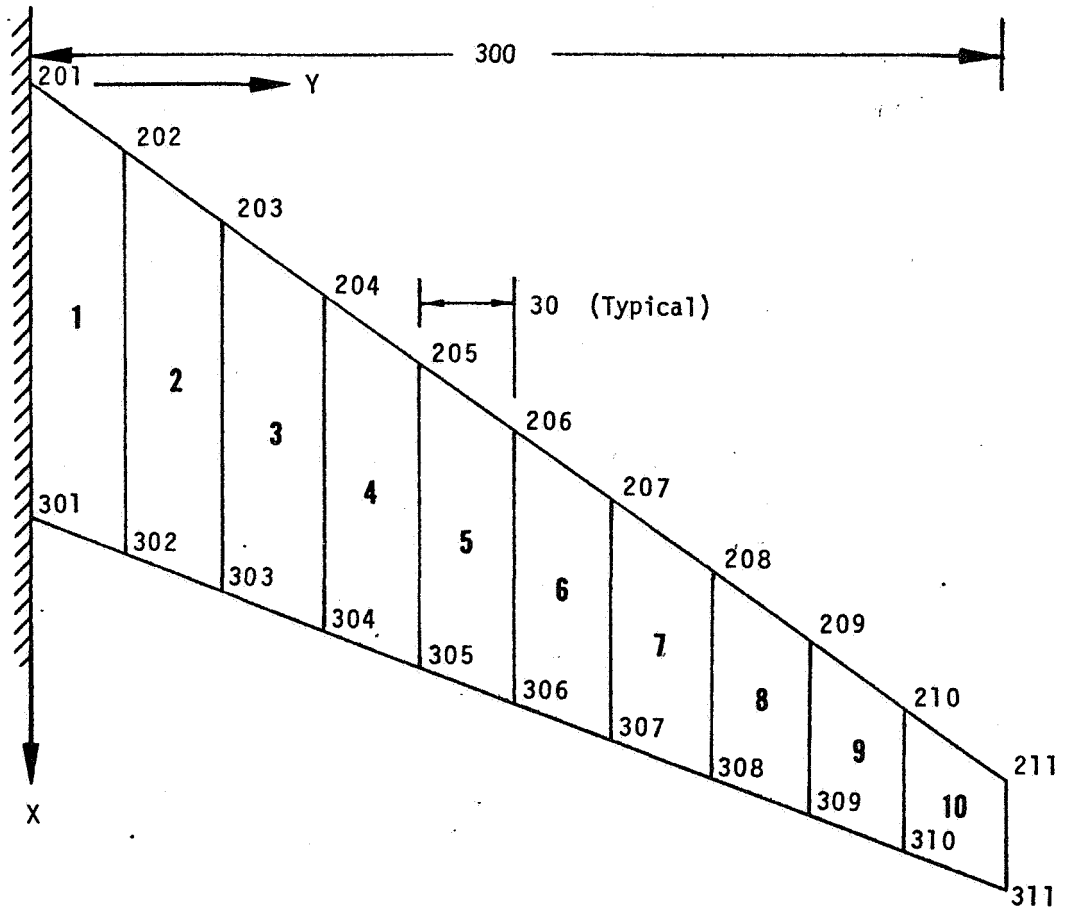


Figure 8-5. Wing Panel Definitions

8.3.2 Execution-Control Data

```
1 BEGIN CONTROL PROGRAM WEIGHT
2 PROBLEM ID (EXAMPLE ATLAS WEIGHTS ANALYSIS)
3 READ INPUT (MODE2)
4 EXECUTE MASS (INERTIAS)
5 PRINT OUTPUT (MASS, STATEMENT, MDC=MDC****)
6 END CONTROL PROGRAM
```

NOTES:

- ① Required first statement in an ATLAS execution-control deck.
- ② Assigns a title to the problem. This title will appear in the printout.
- ③ Directs the system to read and check the model-definition data.
- ④ Directs the system to execute the MASS processor and compute inertias for the defined weight panels
- ⑤ Requests a printout of the weight statement and computed panel weights.
- ⑥ Required last statement in an ATLAS execution-control deck.

8.3.3 Model-Definition Data

```
1 BEGIN NODAL DATA
2 1 160. 0. 0. 15. TO 7 273.2 300. 0. 7.5
3 11 0. *4 17 233.2 **
4 101 160. 0. -45.
5 111 0. 0. -45.
6 201 0. 0. 0. TO 211 233.2 300. 0.
7 301 160. *3 311 273.2 **
8 END NODAL DATA

9 BEGIN STIFFNESS DATA
10 BEGIN ELEMENT DATA
11 SPAR N10 1 2 .25 1.5 TO N15 6 7
12 SPAR N20 11 12 .25 1.5 TO N25 16 17
13 SPAR N30 7 17 .25 1.5
14 COVER N1 1 12 11 .25
15 COVER N2 1 2 13 12 .25 TO N6 5 6 17 16
16 COVER N7 6 7 17 .25
17 BEAM N51 13 111 .25 0. 0. .06 .03 .03
18 BEAM N52 3 101 **
19 END ELEMENT DATA
20 END STIFFNESS DATA
```

```

21 BEGIN SUBSET DEFINITION
22 SUBSETS OF STIFFNESS SET 1
23 E1 = 30
24 E2 = 20 TO 25
25 E3 = 10 TO 15
26 E4 = COVERS
27 E5 = 51
28 E6 = 52
29 END SUBSET DEFINITION

30 BEGIN MASS DATA
31 BEGIN CONDITION DATA
32 PANEL DATA 1 CONDITION 1
33 END CONDITION DATA
34 BEGIN LABEL DATA
35 LEVEL1      ** WING STRUCTURE      #
36 EK1         ** TIP CLOSURE RIB     #
37 LEVEL2      ** FRONT SPAR          #
38 EK2 (W)     ** WEB                  #
39 EK2 (U)     ** UPPER CAP            #
40 EK2 (L)     ** LOWER CAP            #
41 LEVEL2      ** REAR SPAR           #
42 EK3 (W)     ** WEB                  #
43 EK3 (U)     ** UPPER CAP            #
44 EK3 (L)     ** LOWER CAP            #
45 LEVEL2      ** SURFACE MATERIAL    #
46 EK4 (U)     ** UPPER                #
47 EK4 (L)     ** LOWER                #
48 LEVEL2      ** WING STRUTS         #
49 EK5         ** FORWARD              #
50 EK6         ** AFT                  #
51 END LABEL DATA
52 BEGIN PANEL DATA 1
53 1 201 202 302 301 TO 10
54 END PANEL DATA 1
55 END MASS DATA

56 END PROBLEM DATA

```

NOTES:

- ① Identifies the start of the NODAL data
- ② Defines nodes 1,2,...,7 along the rear spar
- ③ Defines nodes 11,12,...,17 along the front spar
- ④,⑤ Define strut nodes 101 and 111
- ⑥,⑦ Define nodes used to define the weight panels

- ⑧ Identifies the end of the NODAL data
- ⑨ Identifies the start of the structural model data
- ⑩ Identifies the start of the structural element data
- ⑪, ⑫ Define the elements along the front and rear spars (element numbers 10-15 and 20-25)
- ⑬ Defines the wing tip closure rib (element number 30)
- ⑭-⑯ Define the wing surface material (element numbers 1-7)
- ⑰, ⑱ Define the wing struts (element numbers 51 and 52)
- ⑲ Identifies the end of the structural element data
- ⑳ Identifies the end of the structural model data
- ㉑ Identifies the start of the definition of element subsets for the weight statement
- ㉒ Specifies elements for the structural model designated SET 1
- ㉓-㉔ Define element subsets for:
- ㉓ tip closure rib
 - ㉔ front spar
 - ㉕ rear spar
 - ㉖ surface material
 - ㉗ forward strut
 - ㉘ aft strut
- ㉙ Identifies the end of the definition of element subsets
- ㉚ Identifies the start of the weights data
- ㉛ Identifies the start of the panel weight CONDITION data
- ㉜ Defines panel definition 1 as weight condition 1
- ㉝ Identifies the end of the panel weight CONDITION data
- ㉞ Identifies the start of the weight statement definition data
- ㉟-㊱ Define the labels for each line of the weight statement and the corresponding element subsets or level of

summation

- ⑤1 Identifies the end of the weight statement definition data
- ⑤2 Identifies the start of the weight PANEL definition data
- ⑤3 Defines weight panels 1-10
- ⑤4 Identifies the end of the weight PANEL definition data
- ⑤5 Identifies the end of the weights data
- ⑤6 Identifies the end of the model-definition data deck

8.3.4 Results

Representative output from the weights analysis is shown on the following pages. Printout of the total weight, panel weights, and weight statement are included.

ATLAS MASS MODULE WEIGHT STATEMENT --- DATA SET 1
EXAMPLE ATLAS WEIGHTS ANALYSIS

ITEM	WEIGHT	XCG	YCG	ZCG
WING-STRUCTURE	2138.06	152.332	127.476	-0.066
TIP CLOSURE RIB	(27.00)	253.200	300.000	0.000
FRONT SPAR	(327.75)	108.151	139.131	0.000
WEB	213.74	103.644	133.333	-0.000
UPPER CAP	57.01	116.600	150.000	11.250
LOWER CAP	57.01	116.600	150.000	-11.250
REAR SPAR	(276.58)	212.499	139.131	0.000
WEB	180.36	210.311	133.333	-0.000
UPPER CAP	48.11	216.600	150.000	11.250
LOWER CAP	48.11	216.600	150.000	-11.250
SURFACE MATERIAL	(1500.47)	149.280	120.000	0.000
UPPER	750.23	149.280	120.000	12.000
LOWER	750.23	149.280	120.000	-12.000
WING STRUTS	(6.26)	103.710	50.000	-22.500
FORWARD	3.36	38.867	50.000	-22.500
AFT	2.90	178.867	50.000	-22.500

NOTES:

1. Output from PRINT OUTPUT (MASS,STATEMENT,-----) statement. Line 5 of Control Program (sec. 8.3.2).
2. The input weight statement labels and order are shown by lines 34-51 in section 8.3.3.

MASS MODULE MATRIX OUTPUT
EXAMPLE ATLAS WEIGHTS ANALYSIS

PAGE 1
78/05/19.

* MASS FILE - MDC001A MATRIX *
* *****

MDC MATRIX
MATRIX NAME 78/05/19. MDC001A
ORDER 10 BY 11 TYPE MIXED CHECKSUM 63070552035123214531
AUXILIARY ID 15012323221606000000 00000000000000000000 00000000000000000000
00000000000000000000 00000000000000000000 00000000000000000000
00000000000000000000 00000000000000000000 00000000000000000000

END OF 10 PRINT AT POSITION 1

PANEL AUX000001 WEIGHT 268.1394
CENTER OF GRAVITY XCG = 14.8199 ZCG = -1.1339
MOMENT OF INERTIA ABOUT C.G. IXX = .7503E+05 IYY = .7388E+06 IZZ = .7040E+06
IXY = -.1163E+05 IXZ = -.1865E+03 IYZ = -.2523E+02

OUTPUT COORDINATE SYSTEM (INERTIAS) GLOBAL

PANEL AUX000002 WEIGHT 248.8201
CENTER OF GRAVITY XCG = 105.0068 YCG = 44.8059 ZCG = -.0934
MOMENT OF INERTIA ABOUT C.G. IXX = .6389E+05 IYY = .5904E+06 IZZ = .5638E+06
IXY = -.1079E+05 IXZ = -.1177E+03 IYZ = -.2718E+02

OUTPUT COORDINATE SYSTEM (INERTIAS) GLOBAL

NOTES:

1. Output from PRINT OUTPUT (MASS,----,MDC=MDC****) statement. Line 5 of Control Program (sec. 8.3.2).
2. The panel weight matrix name and header are printed followed by the panel weight data.
3. The data printed for each panel consist of the panel identification, weight, center of gravity in the global reference system, and the weight moments of inertia about the specified output coordinate system. (GLOBAL by default).

8.4 EXAMPLE 4: VIBRATION ANALYSIS

8.4.1 Description of Model

The model used in this example is the box beam described in section 8.0. The mass distribution is derived solely from the mass and inertia properties of the structural elements in the model.

8.4.2 Description of Analysis

In this example the vibration modes of the wing box are calculated. The box is fully supported at the root.

8.4.3 Execution-Control Data

```
1 BEGIN CONTROL PROGRAM VIBRATION
2 PROBLEM ID (EXAMPLE VIBRATION ANALYSIS)
3 READ INPUT (MODE2)
4 EXECUTE MASS (OPTION=3)
5 PERFORM K-REDUCE
6 EXECUTE VIBRATION (MASS=MDC001A,STIF=KRED,NMODES=8,SUBSETS=N001)
7 PRINT OUTPUT (VIBRATION)
8 EXECUTE EXTRACT (EXNAME=VIBR,LSUB=VMODE,VSET=1,MODE=(1 TO 8),
1   NSUB=N001,BSUB=ON001)
9 EXECUTE GRAPHICS (GNAME=VIBR,TYPE=ORTH,OFFLINE=SC4020,
1   RZ=-40.,RY=25.,VSCALE=100.0,
1   VECTOR3=VMODE,EXNAME=VIBR)
10 END CONTROL PROGRAM
```

NOTES:

- ① Required first statement in an ATLAS execution-control deck
- ② Assigns a title to the problem. This title will appear in the printout.
- ③ Reads and checks the model-definition data deck
- ④ Generates a matrix for mass and inertia properties of the retained freedoms
- ⑤ Generates element stiffnesses and derives a matrix of stiffnesses at the retained freedoms
- ⑥ Performs an eigensolution for 8 modes of vibration, and calculates the generalized mass and generalized stiffness matrices.
- ⑦ Prints the results of the eigensolution

- ⑧ Extracts the results of the eigensolution for plotting
- ⑨ Plots the vibration modes
- ⑩ Required last statement in an ATLAS execution-control deck

8.4.4 Model-Definition Data

```

1 BEGIN NODAL DATA
2 REC SPAN 80. 0. 0., 253.2 300. 0., 80. 0. 100.
3 20 1000. 0. 0.
4 RESUME GLOBAL
5 ANALYSIS FRAME SPAN
6 1 160. 0. 0. 15. TO 7 273.2 300. 0. 7.5
7 11 0. *4 17 233.2 *3
8 101 160. 0. -45.
9 111 0. *2
10 REORDER FROM 20
11 END NODAL DATA

12 BEGIN STIFFNESS DATA
13 BEGIN ELEMENT DATA
14 SPAR N10 1 2 .25 1.5 TO N15 6 7
15 SPAR N20 11 12 *3 N25 16 17
16 SPAR N30 7 17 *2
17 COVER N1 1 12 11 .25
18 COVER N2 1 2 13 12 .25 TO N6 5 6 17 16
19 COVER N7 6 7 17 .25
20 BEAM N51 13 111 .25 0. 0. .06 .03 .03
21 BEAM N52 3 101 **
22 END ELEMENT DATA
23 END STIFFNESS DATA

24 BEGIN BC DATA
25 RETAIN TX TY TZ RX RY FOR 1 TO 100
26 FREE ALL FOR 20
27 SUPPORT ALL FOR 1 11 101 111
28 END BC DATA

29 BEGIN SUBSET DEFINITION
30 SUBSETS OF NODAL SET 1
31 ON001 = 1 2 12 13 3 4 14 15 5 6 16 17 7 +
31 6 5 4 3 2 1 11 12 13 14 15 16 17
32 N001 = 1 2 12 13 3 4 14 15 5 6 16 17 7
33 END SUBSET DEFINITION

34 BEGIN MASS DATA
35 BEGIN CONDITION DATA
36 STAGE 1 CONDITION 1 0 0 0
37 END CONDITION DATA

```

38 END MASS DATA
39 END PROBLEM DATA

NOTES:

- ① Identifies the start of the NODAL data
- ② Defines a local coordinate sysem with y along the center of the structure
- ③ Defines a reference node in the local analysis frame
- ④ Specifies that the next data input will be referenced to the GLOBAL coordinate system
- ⑤ Specifies reference frame SPAN to be used as the analysis frame for the following nodes
- ⑥-⑨ Define nodes 1-7, 11-17, 100, 111
- ⑩ Reorders the nodes for efficient solution
- ⑪ Identifies the end of the NODAL data
- ⑫ Identifies the start of the structural model data
- ⑬-⑫ Define the structural elements
- ⑮ Identifies the end of the structural model
- ⑯ Identifies the start of the boundary condition (BC) data
- ⑰ Retains the freedoms to be used in the vibration analysis. Freedoms 1-7, 11-17 are defined relative to the analysis frame SPAN
- ⑱ Frees node 20
- ⑳ Supports the structure at nodes 1, 11, 101 and 111
- ㉑ Identifies the end of the BC data. Seventy retained freedoms, all in a local analysis frame, have been defined
- ㉒ Identifies the start of SUBSET DEFINITION data. These data are used to define the collections of nodes required for plotting mode shape data.
- ㉓ Identifies the NODAL set from which subsets are created

- ③1 Defines an ordered nodal subset that specifies the lines to be drawn
- ③2 Defines the subset of retained nodes to be used in plotting the mode shapes
- ③3 Identifies the end of the SUBSET DEFINITION data
- ③4 Identifies the start of the MASS data
- ③5 Identifies the start of the mass-distribution CONDITION data
- ③6 Specifies that the element mass and inertia properties are to be used in forming the mass matrix
- ③7 Identifies the end of the CONDITION data
- ③8 Identifies the end of the MASS data
- ③9 Identifies the end of the model-definition data deck

8.4.5 Results

The vibration modes data generated by this analysis are presented in printed and graphical form on the following pages.

THE EQUILIBRIUM OR COMPATIBILITY CHECK GIVEN BELOW IS THE UNBIASED ROOT MEAN SQUARE (RMS) WHICH IS A MEASURE OF THE DEVIATION FROM ZERO OF THE I-TH RESIDUE VECTOR.

TABLE OF EQUILIBRIUM RESULTS

MODE	EIGENVALUE (CUTOFF = .7E-04)	RMS ERROR
1	.718069E+C4	.1182168E-06
2	.2081459E+C5	.4739847E-06
3	.6825651E+C5	.2256754E-06
4	.7323175E+C5	.1991224E-06
5	.1316052E+C6	.2987355E-06
6	.1975317E+C6	.3917757E-06
7	.3120153E+C6	.3427324E-06
8	.3517515E+C6	.3018686E-06

NOTES:

1. Printout generated automatically during execution of the VIBRATION Processor.
Line 6 of Control Program (sec. 8.4.3).
2. Any eigenvalue that is less than the value of CUTOFF is set equal to zero. Good values of the RMS error were obtained for this problem; large values indicate that numerical problems may have been encountered.
3. Each row in this printout corresponds to an eigenvector requested by the key-word NMODES.

THE MODE SHAPE ORTHOGONALITY CHECK, GIVEN BELOW, IS A SYMBOLIC REPRESENTATION OF THE SUB-DIAGONAL LOWER TRIANGULAR PORTION OF PHI-T X MASS X PHI, WHERE PHI IS THE MATRIX OF MODE SHAPES.

SYMBOL	INDICATES
0	TERM) 1.E-12
1	1.E-12) TERM) 1.E-08
2	1.E-08) TERM) 1.E-04
X	1.E-04) TERM

PCW
2 0
3 00
4 000
5 0000
6 00000
7 000000
8 0000000

W A R N I N G --- MODE SHAPE MATRIX (MODES01) ALREADY EXISTS ON VIBRNF, IT IS OVER-WRITTEN.
W A R N I N G --- SUBSET MODE SHAPE MATRIX (SM00101) ALREADY EXISTS ON VIBRNF, IT IS OVER-WRITTEN.

RUN TIME FOR ABOVE OVERLAY 1.495 CP SECONDS

NOTES:

1. Printout generated automatically during execution of the VIBRATION Processor. Line 6 of Control Program (sec. 8.4.3).
2. Values of 0 and 1 are acceptable; 2 indicates possible problems, and X is unacceptable except for rigid body replacement modes.

FREQUENCIES WITH INDEX NAME = FREQS01

MODE	HERTZ
1	.1348663E+02
2	.2296192E+02
3	.4159292E+02
4	.4306950E+02
5	.5773733E+02
6	.7073568E+02
7	.8890130E+02
8	.9439263E+02
9	.1008016E+03
10	.1209494E+03
11	.1277223E+03
12	.1290542E+03
13	.1491678E+03
14	.1607089E+03
15	.1614201E+03
16	.1809811E+03
17	.197238E+03
18	.2181713E+03
19	.2421039E+03
20	.2535039E+03
21	.274731E+03
22	.2826233E+03
23	.2882163E+03
24	.3119086E+03
25	.3300413E+03
26	.3438753E+03
27	.3458625E+03
28	.3622728E+03
29	.3758760E+03
30	.380807E+03
31	.3919441E+03
32	.4017574E+03
33	.4149905E+03
34	.4269076E+03
35	.4392786E+03
36	.4452206E+03
37	.4540321E+03
38	.4590343E+03
39	.4745080E+03
40	.4998074E+03
41	.5210679E+03
42	.5359899E+03
43	.5478690E+03
44	.5651002E+03
45	.5784522E+03
46	.5814845E+03
47	.5959839E+03
48	.6026949E+03
49	.6384366E+03
50	.6775738E+03
51	.7054090E+03
52	.7470472E+03
53	.7575067E+03
54	.7714838E+03
55	.8317394E+03
56	.8908090E+03
57	.9775019E+03
58	.9789446E+03
59	.1069542E+04
60	.1264779E+04

NOTES:

1. Output from PRINT OUTPUT (VIBRATION) statement. Line 7 of Control Program (sec. 8.4.3).
2. The number of frequencies printed is specified by the key-word NFREQS in the EXECUTE VIBRATION statement; line 6 of the Control Program.

SOLUTION MODE SHAPES
WITH INDEX NAME = MODES01

ROW	MODE	1	2	3	4	5	6	7	8
1	1	-.00014391	-.00093896	-.00032224	.04152713	.00022391	-.00032869	-.00043928	.00067473
2	2	.00011610	-.00102504	-.00212944	.07046950	.00085772	.00036151	.00020686	.00071142
3	3	.03240074	.15188939	-.18188038	-.00536352	-.45599851	.17059962	.62495350	-.20181455
4	4	-.00071588	.00366688	.00094170	.00001635	-.00645134	.00711763	.00212285	.01498103
5	5	-.00096032	-.00259241	.00206510	.00024843	.00055414	.00011409	.00067019	.00431026
6	6	-.00038690	-.00171952	.00008903	.07469423	.00056747	-.00091496	.00090735	.00124316
7	7	.00015777	-.00270523	-.00473612	.17837365	-.00179808	.00036111	.00016517	.00104593
8	8	.11424195	.43759132	.43933211	-.01037585	-.68209911	.10346213	.28483758	-.66379993
9	9	-.00140667	.00802953	-.00135538	.00003833	-.00635955	.01317407	-.00731508	.02246816
10	10	.00208184	-.00352345	.00253476	.00000506	.00000566	.00351096	.00366691	.00286920
11	11	-.00338442	-.00156158	-.00043689	.09305568	-.00015292	-.00120015	-.00081786	.00033082
12	12	.00014172	-.00457200	-.00822753	.33370762	-.00381574	-.00011967	.00022753	.00051349
13	13	.26099305	.75569733	-.63141508	-.0012479	-.50421317	-.31728836	.18759366	-.84220083
14	14	-.00184470	.01176017	-.00656640	.00006052	-.00093342	.01290942	-.01376093	.03593787
15	15	-.00328858	-.00234680	-.00068890	.00006704	-.00675052	.00725054	.00090555	.00035348
16	16	-.00037800	-.00131991	-.00038177	.09170971	-.00043284	-.00134476	-.00077171	.00003773
17	17	.00010040	-.00687506	-.01246413	.53205025	-.00677808	-.00098843	.00038376	-.00096201
18	18	.46871598	.97155066	-.50442162	-.00105841	.004640293	-.77120001	.16834482	-.66416298
19	19	.00164014	.01179736	.01084104	.00006593	.00089866	.00472866	-.01292413	.05085428
20	20	.00435740	.00019533	-.00779258	-.00128655	-.00784084	.00181721	.00560819	.00175359
21	21	.00037176	-.00101290	.00009029	.07042610	-.00021908	-.00132039	.00078590	.0007642
22	22	.00006095	-.00886717	-.01732550	.76110796	-.01055937	-.00217500	.00066847	.00388152
23	23	.72247298	1.00000000	.69628248	.00949573	.64717868	-.54117306	.42752088	-.46502941
24	24	-.00030818	.00434353	-.00754197	.00004348	.00459201	.00308311	-.01151914	.05234316
25	25	.00453598	.00200617	-.01447683	-.00017351	-.00643695	-.01049784	.00937178	.00466621
26	26	-.00736534	-.00065923	.00085650	.03669150	.00035938	-.00115033	-.00083682	.00044213
27	27	.00002116	-.01108986	-.02242562	1.00000000	-.01460678	-.00346412	.00099177	-.00616232
28	28	.00000000	.86162346	1.00000000	.02039335	1.00000000	.44270061	1.00000000	-.34970948
29	29	.00245148	-.01217756	.00709686	.00000193	-.00878931	.02086804	.01665125	.03831552
30	30	.00466285	.00303295	-.01410358	-.00018570	-.00728207	-.01211136	.01004172	.00217313
31	31	-.00017246	.00040472	.00041065	.02912868	.00052717	.00042509	.00028094	.0001420
32	32	.00009980	-.00020055	-.00081308	.01790051	-.00038767	.00023084	.00020700	.00012401
33	33	.06234390	.13384257	.47961759	.00311832	.21746080	.84509547	.08440241	-.19811419
34	34	-.00070491	.00111646	.00281529	.0002621	-.00269415	.00288766	.00148704	.00774270
35	35	.00101491	.00113803	.00165702	.00002003	-.00045839	.00208233	.00015820	.00036152
36	36	-.00038597	.00130221	.000354722	-.06215507	.00013064	-.00091525	.00050308	.00006109
37	37	.0021459	-.00085837	.00239247	.06373141	-.00038579	.00054421	.00037094	.00030244
38	38	.19336038	-.36606147	-.84469405	-.00001807	.26583213	.54934623	.07391428	.34265973
39	39	.00156095	.00364819	.00311990	.00004139	-.00578460	.00493970	.00161843	.01313028
40	40	-.00151670	.00244452	-.00076660	.00001604	.00079198	.00700201	-.00105701	-.00110669
41	41	-.00031389	.00153319	.00415572	-.12286572	.00195088	-.00053003	.00060494	.00058397
42	42	.00109916	-.00263978	-.00514232	.18618237	-.00216353	.00039738	.00031365	.00072907
43	43	.39010207	-.61832336	.76299576	-.01030061	-.06277001	-.34099034	.14731485	.88014682
44	44	.00245117	.00963802	-.00164435	.00004202	-.00069399	.01656277	-.00950515	.02321210
45	45	-.00249212	.00076646	-.00538611	.00001408	.00415237	.00293181	.00051622	.00188939
46	46	-.00732552	.00135467	.00467263	-.14539925	.00228723	-.00000000	.00055915	.00160585
47	47		.00507200	.00915105	.37682590	-.00444444		.00027230	.00027230
48	48		-.0078	-.15900838	-.00849949			.730	
49	49			-.01053640	.00004444				

NOTES:

- Output from PRINT OUTPUT (VIBRATION) statement. Line 7 of Control Program (sec. 8.4.3).
- The number of modes printed is specified by the key-word NMODES in the EXECUTE VIBRATION statement; line 6 of the Control Program.
- Each row corresponds to a structural degree of freedom. The printout order corresponds to that of the order of the freedoms for the stiffness and mass matrices.

GENERALIZED MASS
WITH INDEX NAME = GMASS01

MODE	MODE 1	MODE 2	MODE 3	MODE 4	MODE 5	MODE 6	MODE 7	MODE 8
1	-1102234E+01	-5906386E-13	-1132427E-13	3955170E-15	-2153833E-13	-9547918E-14	-2664535E-14	-1554312E-14
2	-5906386E-13	2345088E+01	8060219E-13	-6897261E-14	2742251E-13	3641532E-13	-1898491E-13	9547918E-14
3	-1132427E-13	8060219E-13	2107121E+01	8100465E-13	-3419487E-13	-2886580E-14	-4440892E-14	1199041E-13
4	3555170E-15	-6897261E-14	8100465E-13	8837760E+00	-4135581E-14	-1290634E-14	1595946E-14	-1443290E-14
5	-2153833E-13	2742251E-13	-3419487E-13	-4135581E-14	1514362E+01	-6483702E-13	-1287859E-13	-9547918E-14
6	-9547918E-14	3641532E-13	-2886580E-14	-1290634E-14	6483702E-13	2188789E+01	-7105427E-14	1243450E-13
7	-2664535E-14	-1898491E-13	-4440892E-14	1595946E-14	-1287859E-13	-7105427E-14	1094208E+01	7993606E-14
8	-1554312E-14	9547918E-14	1199041E-13	-1443290E-14	-9547918E-14	1243450E-13	7993606E-14	1767473E+01

GENERALIZED STIFFNESS
WITH INDEX NAME = GSTIF01

MODE	MODE 1	MODE 2	MODE 3	MODE 4	MODE 5	MODE 6	MODE 7	MODE 8
1	7914812E+C4	0.	0.	0.	0.	0.	0.	0.
2	0.	4881299E+05	0.	0.	0.	0.	0.	0.
3	0.	0.	1439090E+06	0.	0.	0.	0.	0.
4	0.	0.	0.	6472046E+05	0.	0.	0.	0.
5	0.	0.	0.	0.	1992979E+06	0.	0.	0.
6	0.	0.	0.	0.	0.	4323553E+06	0.	0.
7	0.	0.	0.	0.	0.	0.	3414096E+06	0.
8	0.	0.	0.	0.	0.	0.	0.	6217114E+06

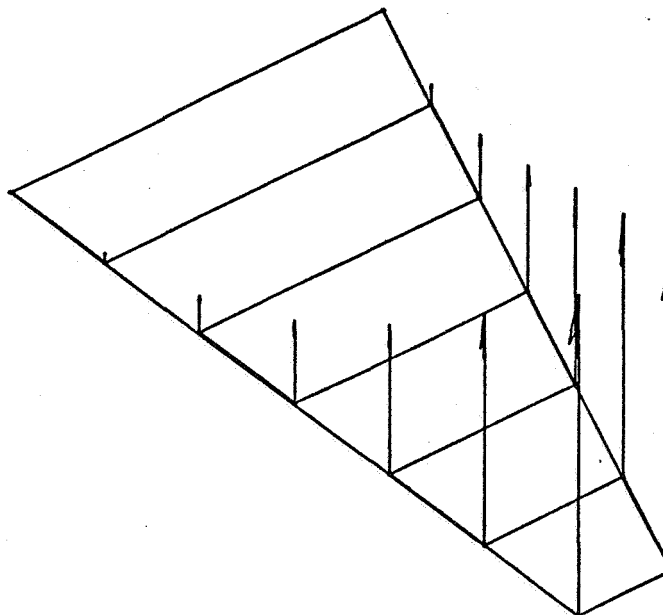
RUN TIME FOR ABOVE OVERLAY .167 CP SECONDS

NOTES:

1. Output from PRINT OUTPUT (VIBRATION) statement. Line 7 of Control Program (sec. 8.4.3).

GNAME = VIBR

PLOT ID = VIBR . MODE SHAPE NO. 1



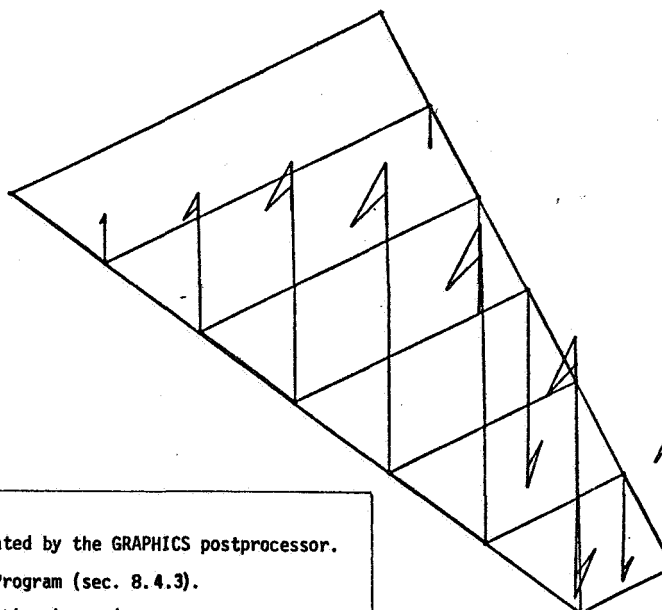
RZ = -40.
RY = 25.
RX = 0.0

OFFLINE
SCALE = 1/43.19
SCALE-X = 1/60.01
SCALE-Y = 1/53.14
SCALE-Z = 1/47.65

EXNAME = VIBR KSET = 1 STAGE = 1
NSUB = N001 FREQUENCY= 13.487LSUB = VMODE

GNAME = VIBR

PLOT ID = VIBR . MODE SHAPE NO. 2



RZ = -40.
RY = 25.
RX = 0.0

OFFLINE
SCALE = 1/43.19
SCALE-X = 1/60.01
SCALE-Y = 1/53.14
SCALE-Z = 1/47.65

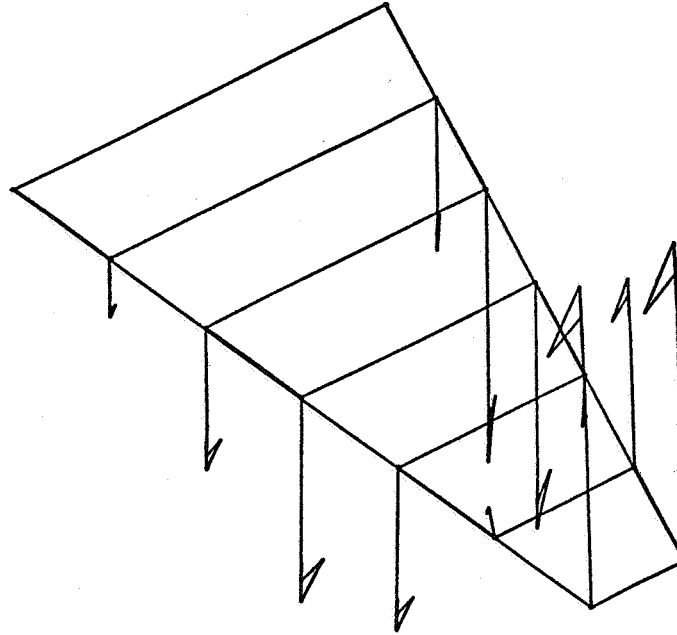
NOTES:

1. SC4020 plots generated by the GRAPHICS postprocessor.
Line 9 of Control Program (sec. 8.4.3).
2. The VECTOR3 plot option is used.
3. The third and fourth modes are shown on the next page.

EXNAME = VIBR KSET = 1 STAGE = 1
NSUB = N001 FREQUENCY= 22.962LSUB = VMODE

GNAME = VIBR

PLOT ID = VIBR . MODE SHAPE NO. 3



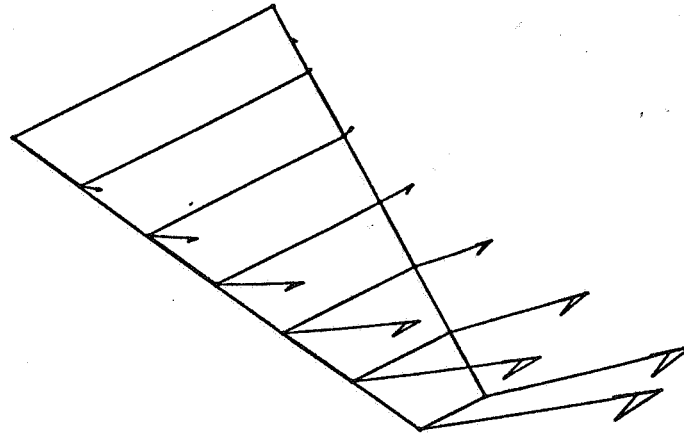
RZ = -40.
RY = 25.
RX = 0.0

OFFLINE
SCALE = 1/43.19
SCALE-X = 1/60.01
SCALE-Y = 1/53.14
SCALE-Z = 1/47.65

EXNAME = VIBR KSET = 1 STAGE = 1
NSUB = N001 FREQUENCY= 41.593LSUB = VMODE

GNAME = VIBR

PLOT ID = VIBR . MODE SHAPE NO. 4



RZ = -40.
RY = 25.
RX = 0.0

OFFLINE
SCALE = 1/49.70
SCALE-X = 1/69.06
SCALE-Y = 1/61.15
SCALE-Z = 1/54.84

EXNAME = VIBR KSET = 1 STAGE = 1
NSUB = N001 FREQUENCY= 43.070LSUB = VMODE

NOTES:

In this mode, motion in the y-direction predominates

This has led to the outline being smaller than in the previous three modes

If it appears that a rotation has also occurred, that is because the vectors are much closer to the y-direction than in the previous plots

8.5 EXAMPLE 5: FLUTTER ANALYSIS

8.5.1 Description of Model

The model used in this analysis is the box beam described in section 8.0. The mass distribution is derived from the mass and inertia properties of the structural elements in the model. The aerodynamic lifting surface is the surface defined by the cover elements. The box beam is "flying" in air at sea level, and contains structural damping equal to between 2 and 3 percent of the stiffness.

8.5.2 Description of Analysis

In this example, the flutter equation of motion is formed for the wing box, fully supported at its root. The aerodynamic model used to obtain the aerodynamic influence coefficients is based on the doublet lattice theory and is shown in figure 8-6. The first six vibration modes were used as the generalized coordinates in calculating generalized mass, stiffness and airforce matrices. A structural damping of 0.02 or 0.03 was assumed for each generalized coordinate.

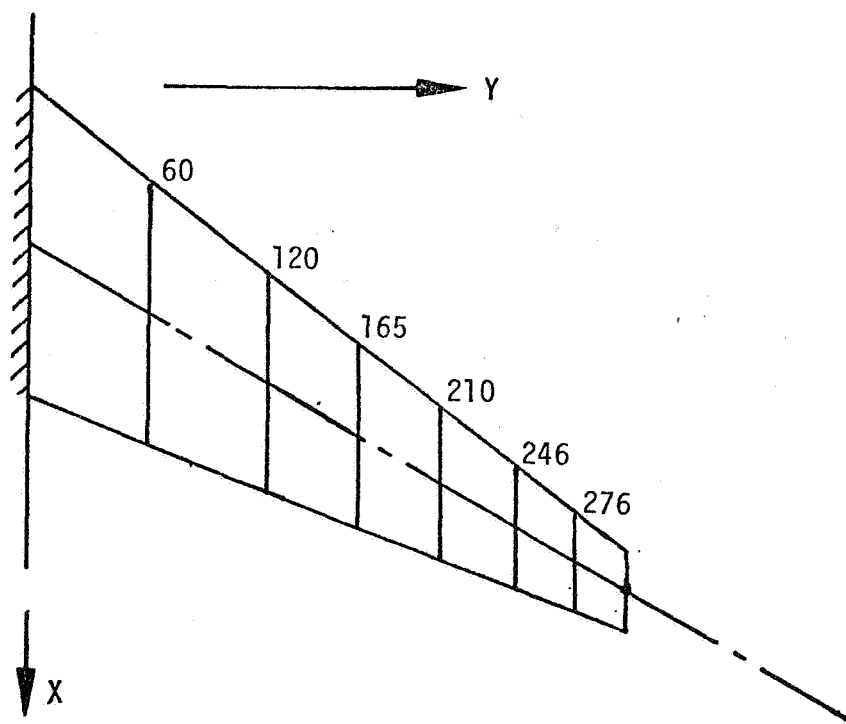


Figure 8-6. Doublet Lattice Box Grid

8.5.3 Execution-Control Data

```
1 BEGIN CONTROL PROGRAM FLUTTER
2 PROBLEM ID (EXAMPLE FLUTTER ANALYSIS)
3 READ INPUT (MODE2)
4 PRINT INPUT (DUBLAT)
5 PRINT INPUT (FLUTTER)
6 EXECUTE INTERP (AIC, N001=BEAMSPLINE, DOF=1110, BEAMIO=(N2,N3))
7 EXECUTE DUBLAT (MACH=0., KVAL=(.1,.05,.025,.01),QUASI=QQ)
8 EXECUTE MASS (OPTION=3)
9 PERFORM K-REDUCE
10 EXECUTE VIBRATION (MASS=MDC001A, STIF=KRED, NMODES=6,
11     SUBSETS=(N1,N2,N3))
12 EXECUTE FLEXAIR (ID=FLEX, DUBLAT, FLUTFREQ=0.0, SUBSET=N1)
13 EXECUTE ADDINT (FLEXAIR=FLEX)
14 EXECUTE FLUTTER (EVAL=1)
15 PRINT OUTPUT (DUBLAT, MACH=0.0, KVAL=.1)
16 PRINT OUTPUT (FLEXAIR, ID=FLEX)
17 PRINT OUTPUT (FLUTTER, EVAL)
18 EXECUTE EXTRACT (LSUB=VGVF, EXNAME=FLUT01)
19 EXECUTE GRAPHICS (GNAME=FLUT01,TYPE=GRAPH,OFFLINE=SC4020,
1 X=V, Y1=G, Y2=F,
2 XMIN=0., XMAX=1000., Y1MIN=-.1, Y1MAX=.05, Y2MIN=0., Y2MAX=60.,
3 EXNAME = FLUT01)
END CONTROL PROGRAM
```

NOTES:

- ① Required first statement in an ATLAS execution control deck
- ② Assigns a title to the problem. This title will appear in the printout
- ③ Reads and checks the model-definition data deck
- ④ Prints the DUBLAT input data
- ⑤ Prints the FLUTTER input data
- ⑥ Generates interpolation coefficients that represent unit displacements at each retained freedom for use in generating AIC matrices
- ⑦ Generates a set of AIC matrices using the doublet lattice aerodynamic theory
- ⑧ Generates a mass matrix
- ⑨ Generates a stiffness matrix

- ⑩ Generates six normal modes of vibration, and generalized mass and stiffness matrices
- ⑪ Generates generalized airforce matrices using the normal modes and AIC matrices
- ⑫ Interpolates the generalized airforce matrices to obtain matrices at a larger number of reduced frequencies
- ⑬ Calculates flutter speeds and eigenvalues at flutter
- ⑭ Prints the AIC matrices
- ⑮ Prints the generalized airforce matrices
- ⑯ Prints the flutter results
- ⑰ Extracts the flutter data for plotting
- ⑱ Plots the V-g and V-f curves
- ⑲ Required last statement in an ATLAS execution control deck

8.5.4 Model-Definition Data

```

1 BEGIN NODAL DATA
  1  160.  0.  0.  15. TO  7  273.2  300.  0.  7.5
 11   0.  *4                17  233.2  *3
101 160.  0. -45.
111   0.  *2
END NODAL DATA

1 BEGIN STIFFNESS DATA
  BEGIN ELEMENT DATA
    SPAR  N10  1  2  .25  1.5 TO  N15  6  7
    SPAR  N20 11 12  *3                N25 16 17
    SPAR  N30  7 17  *2
    COVER N1   1 12 11 .25
    COVER N2   1  2 13 12 .25 TO  N6  5 6  17 16
    COVER N7   6  7 17 .25
    BEAM  N51 13 111 .25 0.  0.  .06  .03  .03
    BEAM  N52  3 101 **
  END ELEMENT DATA
END STIFFNESS DATA

```

```

1 BEGIN BC DATA
    RETAIN TZ FOR 1 TO 17
    RETAIN RX FOR 1 TO 17
    RETAIN RY FOR 1 TO 17
    SUPPORT ALL FOR 1 11 101 111
END BC DATA

1 BEGIN MASS DATA
    BEGIN CONDITION DATA
        STAGE 1 CONDITION 1 0 0 0
    END CONDITION DATA
END MASS DATA

2 BEGIN DUBLAT DATA
3 BEGIN GEOMETRY DATA
4 LIFTING SURFACE DATA
5 PANEL W1 0.0 160.0 233.2 273.2 0.0 300.0 0.0 0.0
6 CHORD DIV 0.0 0.5 1.0
7 SPAN DIV 0.0 0.2 0.4 0.55 0.7 0.82 0.92 1.0
8 END GEOMETRY DATA
9 BEGIN SUBSET DATA
10 SUBSETS OF BOXES
11 SUBSET W1 1 TO 14
12 END SUBSET DATA
13 BEGIN MODAL DATA
14 USE C001 WITH LIFTING SURFACE W1
15 END MODAL DATA
16 END DUBLAT DATA

17 BEGIN FLUTTER DATA
18 DAMP 0.02 0.02 0.02 0.03 0.03 0.03
19 RSET 1 1 2 3 4
20 RSET 2 1 2 3 4 5 6
21 END FLUTTER DATA

1 BEGIN SUBSET DEFINITION
    SUBSETS OF NODAL SET 1
        N001 = 2 3 4 5 6 7 12 13 14 15 16 17
        N002 = 2 TO 7
        N003 = 12 TO 17
        ON001= 2 3 13 14 4 5 15 16 6 7 17 12 2
    END SUBSET DEFINITION

22 END PROBLEM DATA

```

NOTES:

- ① NODAL, STIFFNESS, BOUNDARY CONDITION, MASS, and SUBSET DEFINITION data are specified as for a vibration analysis (see sec. 8.4.3)
- ② Identifies the start of DUBLAT data set
- ③ Identifies the start of DUBLAT planform "GEOMETRY" data
- ④ The data following this record define lifting surfaces as opposed to interference surfaces
- ⑤ Defines the boundary of a trapezoidal surface
- ⑥ Defines the location of box leading and trailing edges
- ⑦ Defines the location of box side edges
- ⑧ Identifies the end of the "GEOMETRY" data
- ⑨ Identifies the start of DUBLAT "SUBSET" data
- ⑩ Specifies that the data following this record define box subsets
- ⑪ Defines a box subset comprised of all 14 boxes on the surface
- ⑫ Identifies the end of the "SUBSET" data
- ⑬ Identifies the start of the DUBLAT "MODAL" data
- ⑭ Specifies that the modal coefficients C001 are to be used to obtain displacements and pitching rotations at the aerodynamic control points defined by the boxes in subset W1
- ⑮ Identifies the end of the "MODAL" data
- ⑯ Identifies the end of the DUBLAT data set
- ⑰ Identifies the start of the FLUTTER data set
- ⑱ Defines the structural damping coefficient associated with each generalized coordinate (normal mode of vibration)

- ①9 Selects the generalized coordinates used to form the flutter equation
- ②0 Selects a different set of generalized coordinates to form a second flutter equation
- ②1 Identifies the end of the FLUTTER data set
- ②2 Identifies the end of the model-definition data deck

8.5.5 Results

The flutter speeds and the resulting v-g and v-f graphs are presented on the following pages together with representative output from the VIBRATION and DUBLAT postprocessors.

EXAMPLE FLUTTER ANALYSIS

```
*****
*   LIFTING SURFACE GEOM.   *
*   *****
```

```
X1 = INBOARD LEADING EDGE X COORDINATE
X2 = INBOARD TRAILING EDGE X COORDINATE
X3 = OUTBOARD LEADING EDGE X COORDINATE
X4 = OUTBOARD TRAILING EDGE X COORDINATE
Y1 = INBOARD Y COORDINATE
Y2 = OUTBOARD Y COORDINATE
Z1 = INBOARD Z COORDINATE
Z2 = OUTBOARD Z COORDINATE
```

PANEL ID	X1	X2	X3	X4	Y1	Y2	Z1	Z2
W1	0.	.16000E+03	.23320E+03	.27320E+03	0.	.30000E+03	0.	0.

PANEL ID	CHORDWISE DIVISIONS (FRACTION OF CHORD)
W1	C.

PANEL ID	SPANWISE DIVISIONS (FRACTION OF SPAN)
W1	0.

NOTES:

1. Output from PRINT INPUT (DUBLAT) statement. Line 4 of Control Program (sec. 8.5.3).
2. The input DUBLAT data are shown by lines 2-16 in section 8.5.4.

INPUT ALTITUDES FOR FLUTTER SEARCH

0.

DIAGONAL MATRIX OF STRUCTURAL DAMPING

MODE	STRUCTURAL DAMPING
1	.20000E-01
2	.20000E-01
3	.20000E-01
4	.30000E-01
5	.30000E-01
6	.30000E-01

PETENTION VECTOR SETS

IDENTIFIERS	DEGREES OF FREEDOM				
1	1	2	3	4	5
2	1	2	3	4	6
RUN TIME FOR ABOVE OVERLAY		.030 CP SECONDS			

NOTES:

1. Output from PRINT INPUT (FLUTTER) statement, Line 5 of Control Program (sec. 8.5.3).
2. The input flutter data are shown by lines 17-21 in section 8.5.4.
3. This printout summarizes the user-specified modifications to the flutter equation.

QUASI-INVERSE MATRICES FOR THE FOLLOWING MACH NUMBERS AND REDUCED FREQUENCIES (BASED UPON A REFERENCE LENGTH OF 1.000) HAVE BEEN WRITTEN TO FILE DUBLRNF

MACH NUMBERS	REDUCED FREQUENCIES
0.00000	.1000E+00 .5000E-01 .2500E-01 .1000E-01

THE MACH NUMBERS AND REDUCED FREQUENCIES FOR WHICH QUASI-INVERSE MATRICES ARE AVAILABLE FOR THIS CONFIGURATION ARE AS FOLLOWS

MACH NUMBERS	REDUCED FREQUENCIES
0.00000	.1000E+00 .5000E-01 .2500E-01 .1000E-01

- NOTES:
1. Printed automatically at the end of execution of the DUBLAT Processor. Line 7 of Control Program (sec. 8.5.3).
 2. This printout summarizes the status of the table of quasi-inverse matrices saved on file DUBLRNF.

EXAMPLE FLUTTER ANALYSIS

 * G E N E R A L I Z E D F O R C E S *
 *

PRINTED OUTPUT = GENERALIZED FORCE MATRIX /
 (DYNAMIC HEAD * 1 ** 3)

MACH NUMBER = 0.
 REDUCED FREQUENCY = .1000E+00

NOTES:

Printed by the DUBLAT Output Postprocessor.
 Because the AIC was selected in the INTERPOLATION
 Processor parameter list, these matrices represent
 the aerodynamic influence coefficients associated
 with each retained freedom in the AIC subset.

DEFL. MODE	MODE 1 *** REAL ***	MODE 1 *** IMAG ***	MODE 2 *** REAL ***	MODE 2 *** IMAG ***	MODE 3 *** REAL ***	MODE 3 *** IMAG ***	MODE 4 *** REAL ***	MODE 4 *** IMAG ***
1	-.108838E+03	.709037E+02	0.	0.	-.106788E+04	-.168149E+04	-.129042E+03	.102591E+03
2	0.	0.	0.	0.	0.	0.	0.	0.
3	0.	0.	0.	0.	0.	0.	0.	0.
4	-.125721E+03	-.102713E+03	0.	0.	-.139275E+04	-.194235E+04	-.127661E+03	-.285096E+03
5	0.	0.	0.	0.	0.	0.	0.	0.
6	0.	0.	0.	0.	0.	0.	0.	0.
7	-.315517E+02	-.450249E+02	0.	0.	-.672889E+03	-.493369E+03	-.335161E+02	-.688455E+02
8	0.	0.	0.	0.	0.	0.	0.	0.
9	0.	0.	0.	0.	0.	0.	0.	0.
10	-.283839E+02	-.246918E+02	0.	0.	-.304440E+03	.498219E+03	-.555335E+02	-.385932E+02
11	0.	0.	0.	0.	0.	0.	0.	0.
12	0.	0.	0.	0.	0.	0.	0.	0.
13	-.373111E+02	-.676249E+02	0.	0.	-.892211E+03	-.585482E+03	-.201285E+02	-.206885E+03
14	0.	0.	0.	0.	0.	0.	0.	0.
15	0.	0.	0.	0.	0.	0.	0.	0.
16	-.625208E+01	-.666063E+01	0.	0.	-.752528E+02	-.149364E+03	-.222763E+02	-.685934E+01
17	0.	0.	0.	0.	0.	0.	0.	0.
18	0.	0.	0.	0.	0.	0.	0.	0.
19	-.666334E+01	-.153651E+02	0.	0.	-.182170E+03	.152259E+03	-.159836E+02	-.185354E+02
20	0.	0.	0.	0.	0.	0.	0.	0.
21	0.	0.	0.	0.	0.	0.	0.	0.
22	-.553294E+00	-.438171E+01	0.	0.	-.480239E+02	-.542021E+02	-.111564E+02	-.547553E+01
23	0.	0.	0.	0.	0.	0.	0.	0.
24	0.	0.	0.	0.	0.	0.	0.	0.
25	-.125185E+01	-.170155E+01	0.	0.	-.189730E+02	.377486E+02	.668907E+01	-.190507E+01
26	0.	0.	0.	0.	0.	0.	0.	0.
27	0.	0.	0.	0.	0.	0.	0.	0.
28	-.162402E-01	-.106674E+01	0.	0.	-.121603E+02	.136030E+02	-.350887E+01	-.131536E+01
29	0.	0.	0.	0.	0.	0.	0.	0.
30	0.	0.	0.	0.	0.	0.	0.	0.
31	-.125672E+00	-.262496E+00	0.	0.	-.366128E+01	-.771652E+01	-.178119E+01	-.440698E+00
32	0.	0.	0.	0.	0.	0.	0.	0.
33	0.	0.	0.	0.	0.	0.	0.	0.
34	-.519305E-01	-.168028E+00	0.	0.	-.209239E+01	-.328857E+01	-.104580E+01	-.290278E+00
35	0.	0.	0.	0.	0.	0.	0.	0.
36	0.	0.	0.	0.	0.	0.	0.	0.

1
 -.320447E+03
 -.525303E+02
 0.
 0.
 .01174E+02

RESIDUAL FLEXIBILITY GENERALIZED FORCE MATRIX

MATRIX NAME = FLEXA A, ALTITUDE = 0. , K-VALUE = .1000E+00

	COLUMN 1		COLUMN 2		COLUMN 3		COLUMN 4	
	REAL	IMAGINARY	REAL	IMAGINARY	REAL	IMAGINARY	REAL	IMAGINARY
1	-.16870747E+04	-.62477285E+03	-.10394724E+04	-.13900840E+03	-.73099202E+03	-.14801563E+04	-.82066974E+03	-.76411807E+03
2	-.32321166E+04	-.23911016E+04	-.23249125E+04	-.13995320E+04	-.39883403E+03	-.76333624E+03	-.91367654E+03	-.53805924E+03
3	-.17438651E+04	-.65867558E+03	-.75930430E+03	-.51298876E+03	-.30578345E-02	-.14112600E+04	-.65938630E+03	-.72634972E+03
4	-.60770352E+03	-.70995666E+03	-.51196434E+03	-.14754048E+03	-.21187814E+03	-.97211919E+03	-.86748459E+02	-.47019714E+03
5	-.26822575E+04	-.51789046E+03	-.24912712E+04	-.99386547E+02	-.11343455E+04	-.89191788E+03	-.11428231E+04	-.57749636E+03
6	-.16359361E+03	-.12580261E+04	-.17351870E+03	-.77248323E+03	-.97603989E+03	-.84492188E+02	-.46523156E+03	-.26123850E+02

	COLUMN 5		COLUMN 6	
	REAL	IMAGINARY	REAL	IMAGINARY
1	-.81013532E+03	-.11342422E+04	-.80656666E+03	-.20616736E+03
2	-.18300683E+03	-.23837815E+03	-.18499023E+04	-.54392958E+02
3	-.26359234E+03	-.73496937E+03	-.93433804E+03	-.73431601E+02
4	-.44114889E+03	-.86584380E+03	-.33501666E+02	-.13934874E+03
5	-.96193585E+03	-.86625146E+03	-.13533948E+04	-.21950598E+03
6	-.81678855E+03	-.37205716E+03	-.58362211E+02	-.12425165E+03

NOTES:

1. Output from PRINT OUTPUT (FLEXAIR) statement. Line 15 of Control Program (sec. 8.5.3).
2. This matrix is obtained from the AIC matrix using $[Q] = [MODES]^T [AIC] [MODES]$

INPUT DATA AND OPTIONS SUPPLIED TO THE FLUTTER TECHNICAL MODULE

NO.OF DEGREES OF FREEDOM----- 4
MACH NUMBER----- 0.000
PRINT LEVEL-----1
MATCHED POINT SOLUTION REQUESTED-----FALSE
NO. OF ALTITUDES-----1
NO. OF LAGUERRE ITERATIONS----- 1
STILL AIR MODES REQUESTED-----FALSE
UNITS SYSTEM-----BRITISH
V-G PLOTS REQUESTED-----TRUE
GENERALIZED MASS MATRIX NAME-----GMASS01
GENERALISED STIFFNESS MATRIX NAME-----GSTIF01
GENERALISED AIRFORCE MATRIX NAME-----GAF
NO. OF REDUCED FREQUENCIES----- 61
FLUTTER SEARCH WINDOW
VMIN -----.00000
VMAX -----.10000+101
FMIN ----- 0.
FMAX -----.10000E+04

NOTES:

1. Output from PRINT OUTPUT (FLUTTER) statement. Line 16 of Control Program (sec. 8.5.3).

FLUTTER ROOTS

CONDITION 1
 CHANGES 1
 RETENTION VECTOR SET 1
 DEGREES OF FREEDOM 1
 ALTITUDE 2 3 4
 AIRSPEED 0.0 FEET
 MASS RATIO 0.000 KNOTS TAS
 CROSSINGS FOUND .80132E+06
 DAMPING LEVEL G = 4
 0.

MODE	REDUCED FREQUENCY	FLUTTER SPEED INDEX	FREQUENCY (HERTZ)	FLUTTER SPEED KNOTS TAS	DG/DRF	DFS1/DG
1	.861169E-01	1.10474	13.5541	48.7951	-1.53379	.114258E-03
4	.761586E-01	2.12049	23.0201	93.6599	-1.49549	-.865891E-03
1	.307637E-01	9.49268	41.6056	419.282	-2.14423	-.358210E-02
2	.122079E-01	33.5813	58.4068	1483.25	-7.03925	-.173158E-02

NOTES:

1. Printed by the FLUTTER Output Postprocessor. Line 16 of Control Program (sec. 8.5.3).
2. The MASS RATIO is based on the (1,1) element of the generalized mass matrix
3. The mode number is not related to the order of the modes as used in the analysis.
 DG/DRF is the rate of change of damping with reduced frequency. Negative indicates that the mode has become unstable with increasing speed. DFS1/DG is proportional to the rate of change of flutter speed with damping.
4. The repetition of MODE1 with the same sign of DG/DRF indicates that "mode switching" has occurred.

FLUTTER SOLUTION OUTPUT DATA
EXAMPLE FLUTTER ANALYSIS
DEFAULT

CASE 1

DATE 78/04/03

PAGE 3

CONDITION 1
CHANGESET 1
RETENTION VECTOR SET 1
ALTITUDE 0.0

EIGEN-VALUES AT REDUCED FREQUENCY -100E+00

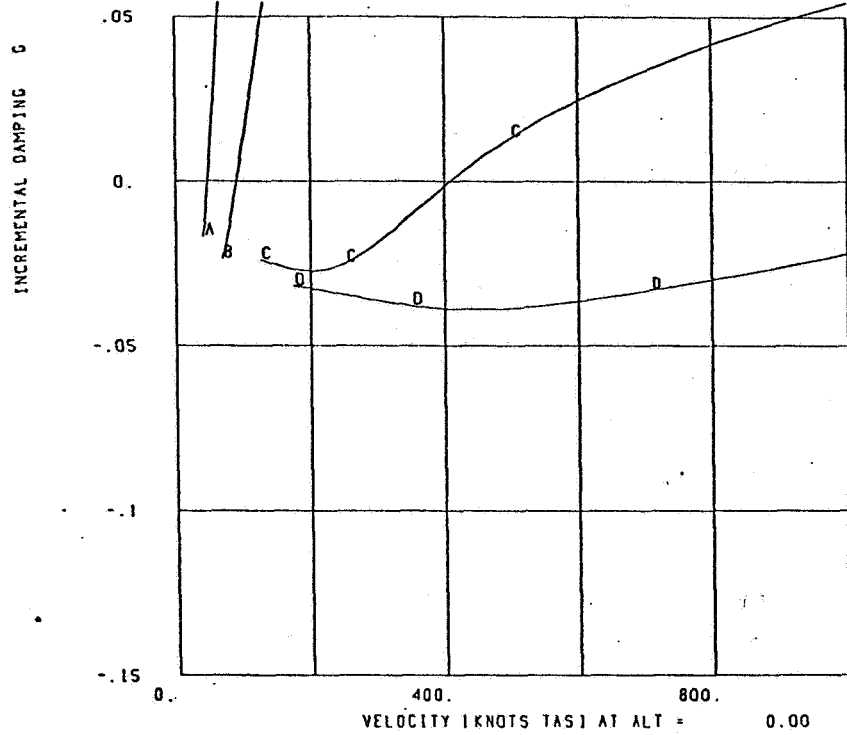
ROOT NO	FLUTTER FREQUENCY (HZ)	DAMPING COEFFICIENT (G)	FLUTTER SPEED (KTS TAS)	COMPLEX ROOT REAL PART	COMPLEX ROOT IMAGINARY PART
1	13.548	-.01670	42.0	.13800E-03	-.23050E-05
2	23.034	-.02348	71.4	.47741E-04	-.11209E-05
3	41.603	-.02389	129.0	.14635E-04	-.34968E-06
4	57.753	-.03179	179.0	.75945E-05	-.24143E-06

NOTES:

1. Output from PRINT OUTPUT (FLUTTER) statement. Line 16 of Control Program (sec. 8.5.3).

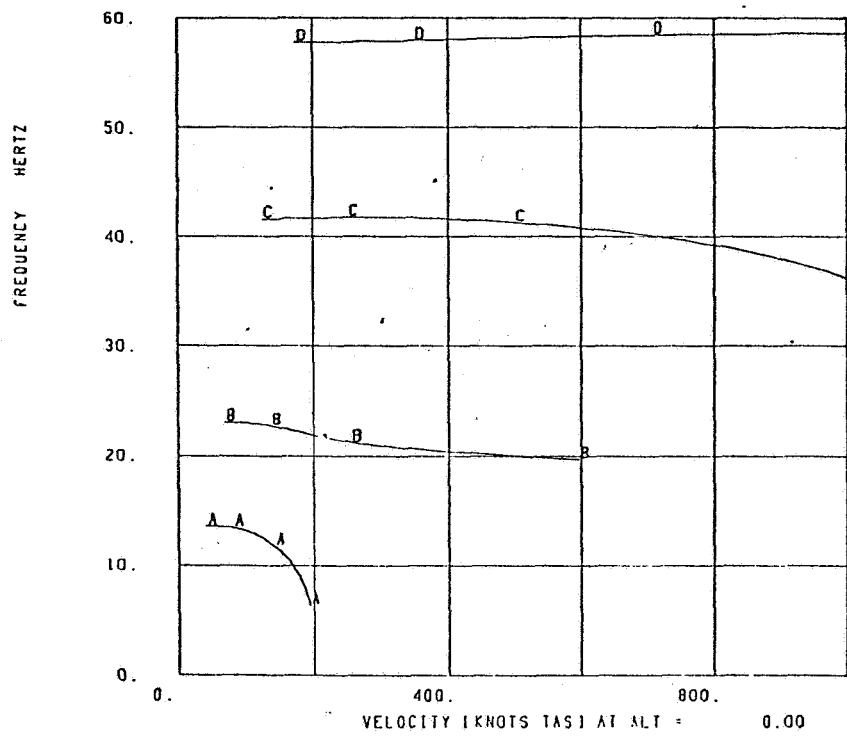
NAME = FLUT01

PLOT 10 = V-G. CASE= 1.CO= 1.CS= 1.RS= 1



NAME = FLUT01

PLOT 10 = V-F. CASE= 1.CO= 1.CS= 1.RS= 1



Flutter Solution V-g and V-f Plots

8.6 EXAMPLE 6: SUBSTRUCTURED STRESS ANALYSIS

8.6.1 Description of Model

The total structural model used in this example is the box beam described in section 8.0. This total structural model is analyzed using two substructures as illustrated in figure 8-7.

8.6.2 Description of Analysis

Substructure 1 is fully supported at the root. The following loads are applied to both substructures:

- A distributed pressure load acting in the GLOBAL Z direction. The pressure is uniform over each element and varies from a maximum at the root to a minimum at the tip.
- A thermal loading consisting of a uniform ΔT of 200°F on the upper surface and -200°F on the lower surface.

In addition, a concentrated tip load acting in the GLOBAL Z direction and divided equally between front and rear spars is applied to substructure 2.

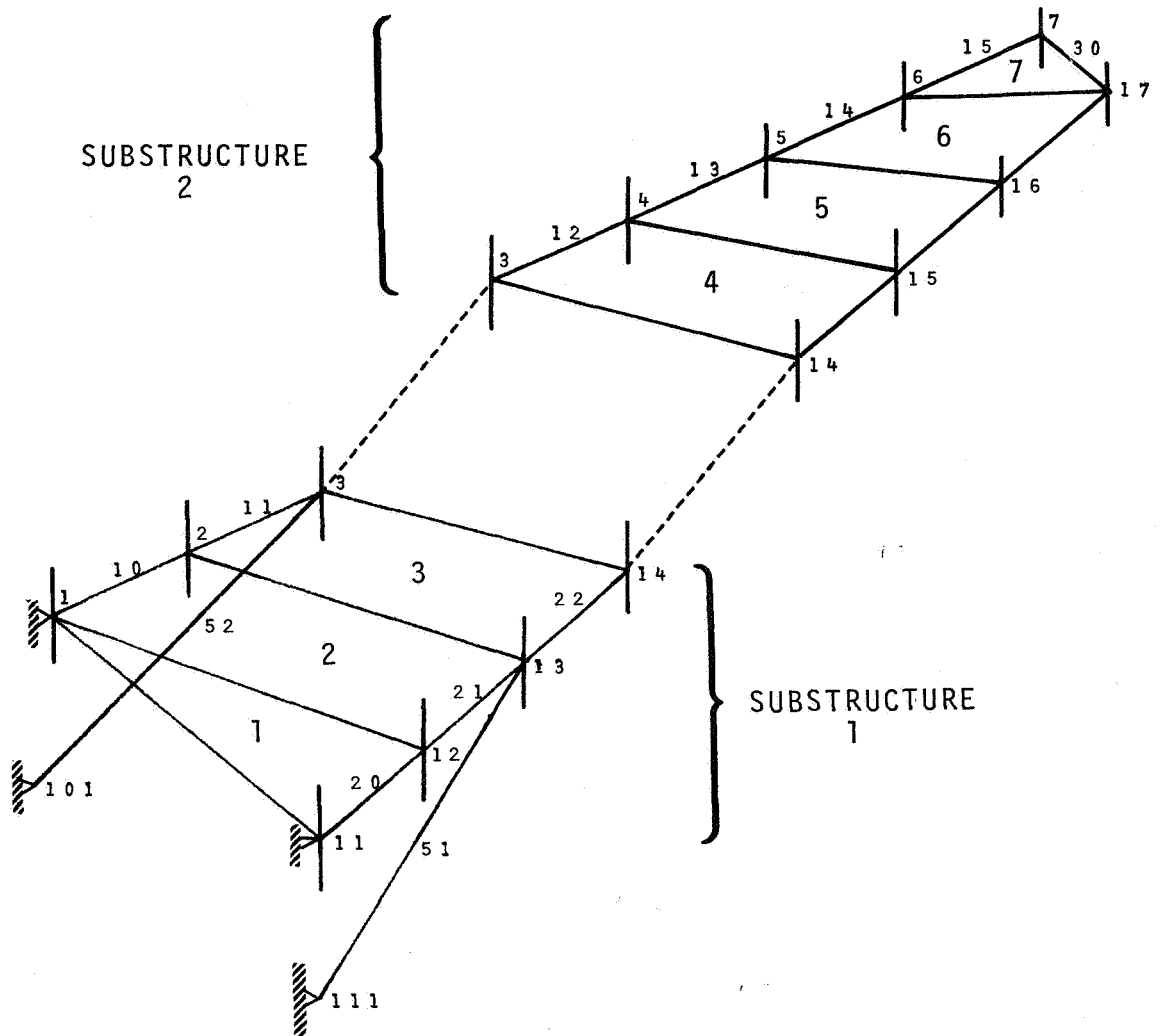


Figure 8-7. Substructured Wing Box Structural Model

8.6.3 Execution-Control Data

```
1 BEGIN CONTROL PROGRAM SUBSTRC
2 PROBLEM ID (EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS)
3 USER COMMON (I)
4 READ INPUT (MODE2)
5 DO 10 I=1,2
6 PRINT INPUT (NODAL,SET=I)
7 PRINT INPUT (STIFFNESS,SET=I)
8 PRINT INPUT (LOADS,SET=I)
9 EXECUTE STIFFNESS (SET=I)
10 CONTINUE
11 READ INPUT (MODE2)
12 PRINT INPUT (INTERACT,SS=(1,2),NODE,CONN,RETA,BC,LOAD)
13 EXECUTE LOADS (SS=(1,2))
14 PERFORM SS-MERGE (STIF,LOADS,SS=(1,2))
15 PERFORM SS-REDUCE (STIF,LOADS,SS=(1,2))
16 PERFORM SS-MERGE (STIF,LOADS,SS=3)
17 PERFORM SS-SSOLVE (SS=3)
18 PERFORM SS-PARTITION (SS=3)
19 PERFORM SS-BACK (SS=(1,2))
20 DO 20 I=1,2
21 EXECUTE STRESS (SS=I)
22 CONTINUE
23 DO 30 I=1,2
24 PRINT OUTPUT (DISP,SS=I)
25 PRINT OUTPUT (STRESS,SS=I)
26 PRINT OUTPUT (LOADS,SS=I)
27 PRINT OUTPUT (REACTIONS,SS=I)
28 CONTINUE
29 END CONTROL PROGRAM
```

NOTES:

- ① Required first statement in an ATLAS execution-control deck. The name selected for this Control Program is SUBSTRC
- ② Assigns a title to the problem. This title will appear in the printout.
- ③ Defines the FORTRAN variable I to be stored in a common block so that the value assigned to I by this Control Program is not destroyed between successive ATLAS statements.
- ④ Directs ATLAS to read and check the first model-definition data deck. The substructure interact data forms the second data deck.

- (5-8) Request formatted printout of the following input data for structural model SET numbers 1 and 2:
- (6) nodes
 - (7) stiffness elements
 - (8) loads
- (9-10) Directs ATLAS to execute the STIFFNESS processor for structural model SET numbers 1 and 2 for calculation of element stiffness matrices
- (11) Directs ATLAS to read and check the second data deck in this job. This data deck, consisting of only the INTERACT data, are input after the STIFFNESS processor is executed so that any inactive freedoms on the substructure interfaces are treated accordingly during use of the automatic retention criteria (see sec. 5.9).
- (12) Requests a formatted printout of the substructure INTERACT data for substructures 1 and 2
- (13) Directs ATLAS to execute the LOADS Processor for substructure (SS) numbers 1 and 2
- (14,15) Direct ATLAS to use the execution-control procedures SS-MERGE and SS-REDUCE to create the gross stiffness and loads matrices for substructures 1 and 2, and then create the corresponding reduced matrices for subsequent interaction
- (16,17) Direct ATLAS to use the execution-control procedures SS-MERGE and SS-SSOLVE to create the gross stiffness and loads matrices for the highest-level substructure 3, and then solve the resulting equations for the unknown displacements
- (18) Directs ATLAS to use the execution-control procedure SS-PARTITION to prepare the loads-dependent matrices associated with substructure 3 for subsequent back-substitution
- (19) Directs ATLAS to use the execution-control procedure SS-BACK for back-substitution in calculation of the unknown displacements for substructures 1 and 2
- (20-22) Direct ATLAS to execute the STRESS processor for calculation for element stresses associated with substructures 1 and 2

(23)-(28)

Request formatted printout of the following calculated data for substructures 1 and 2:

- (25) nodal displacements
- (26) element stresses
- (27) cumulated equivalent nodal loads and resultants of applied loads
- (28) reactions

(29)

Required last statement in an ATLAS execution-control deck

8.6.4 Model-Definition Data

```
1 BEGIN NODAL DATA
2   SET 1
3     REC SPAN 80. 0. 0., 253.2 300.0 0., 80. 0. 100.
4     20 1000. 0. 0.
5     RESUME GLOBAL
6     ANALYSIS FRAME SPAN
7       1 160. 0. 0. 15. TO 3 197.73333 100. 0. 12.5
8       11 0. 0. 0. 15. TO 14 116.6 150. 0. 11.25
9       101 160. 0. -45.
10      111 0. 0. -45.
11      REORDER FROM 20
12   SET 2
13     REC SPAN 80. 0. 0., 253.2 300. 0., 80. 0. 100.
14     21 1001. 0. 0.
15     RESUME GLOBAL
16     ANALYSIS FRAME SPAN
17       3 197.73333 100. 0. 12.5 TO 7 273.2 300. 0. 7.5
18       14 116.6 150. 0. 11.25 TO 17 233.2 300. 0. 7.5
19     REORDER FROM 21
20 END NODAL DATA

21 BEGIN STIFFNESS DATA
22   SET 1
23     BEGIN ELEMENT DATA
24       SPAR N10 1 2 .25 1.5
25       SPAR N11 2 3 **
26       SPAR N20 11 12 .25 1.5 TO N22 13 14
27       COVER N1 1 12 11 .25
28       COVER N2 1 2 13 12 .25 TO N3 2 3 14 13
29       BEAM N51 13 111 .25 0. 0. .06 .03 .03
30       BEAM N52 3 101 **
31     END ELEMENT DATA
32   SET 2
33     BEGIN ELEMENT DATA
34       SPAR N12 3 4 .25 1.5 TO N15 6 7
35       SPAR N23 14 15 .25 1.5 TO N25 16 17
```

```

36      SPAR N30  7 17 .25 1.5
37      COVER N4  3 4  15 14 .25 TO N6 5 6 17 16
38      COVER N7  6 7  17 .25
39      END ELEMENT DATA
40      END STIFFNESS DATA

41      BEGIN BC DATA
42          SET 1 STAGE 1
43              SUPPORT ALL FOR 1 1 101 111
44              SUPPORT RZ FOR 14
45          SET 2 STAGE 1
46              SUPPORT RZ FOR 14
47      END BC DATA

48      BEGIN LOADS DATA
49          SET 1 STAGE 1
50              LOAD CASE ID AIRLOAD **ELEMENT PRESSURE LOADING*
51              LOAD CASE ID THERMAL **200 DEG. UPR. AND -200 LWR. *
52              BEGIN ELEMENT LOAD DATA
53                  DIRECTION GLOBAL 0. 0. 1.
54                  CASE AIRLOAD
55                      1 3.5
56                      **2 1 -.5
57              END ELEMENT LOAD DATA
58              BEGIN ELEMENT THERMAL LOAD DATA
59                  CASE THERMAL
60                      10 11 20 21 22 200. -200. 200. -200.
61                      1 TO 3 200. -200.
62              END ELEMENT THERMAL LOAD
63          SET 2 STAGE 1
64              LOAD CASE ID AIRLOAD **ELEMENT PRESSURE LOADING*
65              LOAD CASE ID TIPLOAD **2000 LB. LOAD AT TIP*
66              LOAD CASE ID THERMAL **200 DEG. UPR. AND -200 LWR. *
67              BEGIN NODAL LOAD DATA
68                  CASE TIPLOAD
69                      7 17 FZ -1000.
70              END NODAL LOAD DATA
71              BEGIN ELEMENT LOAD DATA
72                  DIRECTION GLOBAL 0. 0. 1.
73                  CASE AIRLOAD
74                      4 2.0
75                      **3 1 -.5
76              END ELEMENT LOAD DATA
77              BEGIN ELEMENT THERMAL LOAD DATA
78                  CASE THERMAL
79                      12 TO 15, 23 TO 25, 30 200. -200. 200. -200.
80                      4 TO 7 200. -200.
81              END ELEMENT THERMAL LOAD DATA
82      END LOADS DATA

```

```

83| END PROBLEM DATA
84| BEGIN INTERACT DATA
85|   UNIQUE NODE NUMBERS
86|     DEFINE SS 1 AS SET 1 STAGE 1
87|     DEFINE SS 2 AS SET 2 STAGE 1
88|     SS 3
89|       TOLERANCE .001
90|       INTERACT 1 2
91|       DEFINE HIGHEST SS 3
92| END INTERACT DATA
93| END PROBLEM DATA

```

NOTES:

- ① Identifies the start of the "NODAL" data set
- ②-②① Define the NODAL data set for structural model SET numbers 1 and 2; similar data records are described for the EXAMPLE 1 problem (the same nodes for the non-substructured wing model)
- ②①-④① Define the STIFFNESS data set for structural model SET numbers 1 and 2; similar data records are described for the EXAMPLE 1 problem (the same elements for the non-substructured wing model)
- ④①-④⑦ Define the boundary condition (BC) data set for structural model SET numbers 1 and 2; reference the EXAMPLE 1 problem description.
- ④⑧-⑧② Define the LOADS data set for structural model SET numbers 1 and 2; reference the EXAMPLE 1 problem description of the three loadcases
- ⑧③ Identifies the end of the first model-definition data deck. The first READ INPUT statement reads through this data record.
- ⑧④ Identifies the start of the INTERACT data set. The second READ INPUT statement begins reading with this data record.
- ⑧⑤ Specifies that unique node numbers have been used throughout the substructured model
- ⑧⑥-⑧⑦ Define substructures 1 and 2 to be equivalent to SET=1, STAGE=1 and SET=2, STAGE=1, respectively

- ⑧⑧-⑨⑦ Define substructure 3 to be created by interacting substructures 1 and 2; the geometric nodal coordinate tolerance to be used in automatically identifying the interacting nodes is set to .001
- ⑨① Defines substructure 3 as the highest level substructure in the interact diagram
- ⑨② Identifies the end of the INTERACT data set
- ⑨③ Identifies the end of the second model-definition data deck

8.6.5 Results

Representative output from a substructured stress analysis is shown on the following pages. Printout of input data (nodal, stiffness, and interact) and output data (loads, displacements, stresses, and reactions) are included.

***** NODAL DATA ***** DATA SET NUMBER = 1
 PROBLEM ID -- EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

USER NODE	INPUT RECORD	INTERNAL NODE	X COORD	Y COORD	Z COORD	DX COORD	DY COORD	DZ COORD	NODE REF. FRAME	ANALYSIS REF. FRAME
1	7	6	160.0000	0.0000	0.0000	0.0000	0.0000	15.0000	SPAN	SPAN
2	7	4	178.8667	50.0000	0.0000	0.0000	0.0000	13.7500	SPAN	SPAN
3	7	3	197.7333	100.0000	0.0000	0.0000	0.0000	12.5000	SPAN	SPAN
11	8	9	0.0000	0.0000	0.0000	0.0000	0.0000	15.0000	SPAN	SPAN
12	8	8	38.8667	50.0000	0.0000	0.0000	0.0000	13.7500	SPAN	SPAN
13	8	5	77.7333	100.0000	0.0000	0.0000	0.0000	12.5000	SPAN	SPAN
14	8	2	116.6000	150.0000	0.0000	0.0000	0.0000	11.2500	SPAN	SPAN
20	4	1	579.9890	866.0318	0.0000	0.0000	0.0000	0.0000	SPAN	SPAN
101	9	7	160.0000	0.0000	-45.0000	0.0000	0.0000	0.0000	SPAN	SPAN
111	10	10	0.0000	0.0000	-45.0000	0.0000	0.0000	0.0000	SPAN	SPAN

RUN TIME FOR ABOVE OVERLAY .102 CP SECONDS

NOTES:

1. Output from PRINT INPUT (NODAL) statement. Line 6 of Control Program (sec. 8.6.3).
2. The input nodal data are shown by lines 1-11 in section 8.6.4.
3. Default printout order is by increasing user-assigned node numbers.
4. All coordinates are printed relative to the GLOBAL reference frame.
5. A blank in the "reference frame" columns denotes the GLOBAL reference frame.

USER ID

21	SPAR	INPUT RECORD USER NODES 12 13	26, INTERNAL NO. 8 5	7, INTERNAL NO. 8 5	70, TEMPERATURE PROPERTIES T-WER = .25000 FAREA1U = 1.50000 FAREA1L = 1.50000 FAREA2U = 1.50000 FAREA2L = 1.50000 O111U = 0.00000 O111L = 0.00000 O121U = 0.00000 O121L = 0.00000 A-LMP1U = 1.14583 A-LMP1L = 1.14583 A-LMP2U = 1.04167 A-LMP2L = 1.04167	MATERIAL CODE 1
----	------	--	-------------------------------	------------------------------	--	-----------------

22	SPAR	INPUT RECORD USER NODES 13 14	26, INTERNAL NO. 5 2	2, INTERNAL NO. 5 2	70, TEMPERATURE PROPERTIES T-WER = .25000 FAREA1U = 1.50000 FAREA1L = 1.50000 FAREA2U = 1.50000 FAREA2L = 1.50000 O111U = 0.00000 O111L = 0.00000 O121U = 0.00000 O121L = 0.00000 A-LMP1U = 1.04167 A-LMP1L = 1.04167 A-LMP2U = .93750 A-LMP2L = .93750	MATERIAL CODE 1
----	------	--	-------------------------------	------------------------------	--	-----------------

51	BEAM	INPUT RECORD USER NODES 13 111	29, INTERNAL NO. 5 10	8, INTERNAL NO. 5 10	70, TEMPERATURE PROPERTIES A(1) = .25000 A-VY(1) = 0.00000 A-VZ(1) = 0.00000 J(1) = .06000 IY(1) = .03000 IZ(1) = .03000 A(2) = .25000 A-VY(2) = 0.00000 A-VZ(2) = 0.00000 J(2) = .06000 IY(2) = .03000 IZ(2) = .03000 CONSTR = 0.	MATERIAL CODE 1
----	------	---	--------------------------------	-------------------------------	---	-----------------

NOTES:

1. Output from PRINT INPUT (STIFFNESS) statement. Line 7 of Control Program (sec. 8.6.3).
2. The input stiffness data are shown by lines 21 - 31 in section 8.6.4.
3. Default printout order is by increasing user-assigned element numbers.

PAGE 1

DATE - 77/04/20.

STAGF NO. 1

•
•
•

SUBSTRUCTURED STRESS ANALYSIS

REF. FRAME OR SURFACE	DIRECTION VECTOR OR ORIENTATION MODE
GLOBAL	0.000000 0.000000 1.000000
GLOBAL	0.000000 0.000000 1.000000
GLOBAL	0.000000 0.000000 1.000000

NOTES:

1. Output from PRINT INPUT: (LOADS) statement. Line 8 of Control Program (sec. 8.6.3).
2. The input element loads data are shown by lines 52-57 in section 8.6.4.
3. Printout of the input element-distributed loads is in an increasing element-number order.
4. Only the pressure intensities that are specified for the points corresponding to the element types are printed.
5. Multiple loadcases with distributed loads would all be shown for each element.

DATE - 77/04/20.

SET NO. 1 STAGE NO. 1

*** ELEMENT THERMAL LOADS ***

PROBLEM ID - EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

1	COVER	MATL 1	ELEM TEMP 70	NCDES 1	12	11	P2L	P3L	P4L
	LOADCASE	P1U	P2U	P3U	P4U				
	THERMAL	200.00000	200.00000	200.00000	-200.00000	-200.00000	-200.00000	-200.00000	
2	COVER	MATL 1	ELEM TEMP 70	NCDES 1	2	13	12	P2L	P4L
	LOADCASE	P1U	P2U	P3U	P4U				
	THERMAL	200.00000	200.00000	200.00000	200.00000	-200.00000	-200.00000	-200.00000	-200.00000
3	COVER	MATL 1	ELEM TEMP 70	NCDES 2	3	14	13	P2L	P4L
	LOADCASE	P1U	P2U	P3U	P4U				
	THERMAL	200.00000	200.00000	200.00000	200.00000	-200.00000	-200.00000	-200.00000	-200.00000
10	SPAR	MATL 1	ELEM TEMP 70	NCDES 1	2				
	LOADCASE	N1(U)	N1(L)	N2(U)	N2(L)				
	THERMAL	200.00000	-200.00000	200.00000	-200.00000	0.0000000			
11	SPAR	MATL 1	ELEM TEMP 70	NCDES 2	3				
	LOADCASE	N1(U)	N1(L)	N2(U)	N2(L)				
	THERMAL	200.00000	-200.00000	200.00000	-200.00000	0.0000000			
20	SPAR	MATL 1	ELEM TEMP 70	NCDES 11	12				
	LOADCASE	N1(U)	N1(L)	N2(U)	N2(L)				
	THERMAL	200.00000	-200.00000	200.00000	-200.00000	0.0000000			
21	SPAR	MATL 1	ELEM TEMP 70	NCDES 12	13				
	LOADCASE	N1(U)	N1(L)	N2(U)	N2(L)				
	THERMAL	200.00000	-200.00000	200.00000	-200.00000	0.0000000			
22	SPAR	MATL 1	ELEM TEMP 70	NCDES 13	14				
	LOADCASE	N1(U)	N1(L)	N2(U)	N2(L)				
	THERMAL	200.00000	-200.00000	200.00000	-200.00000	0.0000000			

RUN TIME FOR ABOVE OVERLAY .381 CP SECONDS

NOTES:

1. Output from PRINT INPUT (LOADS) statement. Line 8 of Control Program (sec. 8.6.3).
2. The input element thermal loads data are shown by lines 58-62 in section 8.6.4.
3. Similar format as shown on the preceding page for the input element distributed loads. In this printout, however, the nodal delta-temperatures are shown.

SUBSTRUCTURE NUMBER 1
 PROBLEM ID - EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

*** SUBSTRUCTURE NODAL DATA ***

USER NODE NUMBER	- X -	- Y -	- Z -	ORIGINATING SUBSTRUCTURE NUMBER	ORIGINATING USER NODE NUMBER
20	579.989	866.032	0.000	1	20
14	116.600	150.000	0.000	1	14
3	197.733	100.000	0.000	1	3
2	178.867	50.000	0.000	1	2
13	77.733	100.000	0.000	1	13
1	160.000	0.000	0.000	1	1
101	160.000	0.000	-45.000	1	101
12	38.867	50.000	0.000	1	12
11	0.000	0.000	0.000	1	11
111	0.000	0.000	-45.000	1	111

DATE 77/04/20 PAGE 1

NOTES:

1. Output from PRINT INPUT (INTERACT, NODE) statement, Line 12 of Control Program (sec. 8.6.3).

*** BOUNDARY CONDITION DATA ***

DATE 77/04/20 PAGE 1

SUBSTRUCTURE NUMBER 1
 PROBLEM 10 - EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

FREEDOM ACTIVITIES ARE DENOTED BY THE FOLLOWING

CHARACTER F DENOTES A PARTITION 1 FREEDOM (F=FREE)
 CHARACTER R DENOTES A PARTITION 2 FREEDOM (R=RETAIN)
 CHARACTER S DENOTES A PARTITION 3 FREEDOM (S=SUPPORT)
 A BLANK DENOTES A FREEDOM WITH ZERO STIFFNESS

USER NODE NUMBER	NODAL COORDINATES			ORIGINATING SUBSTRUCTURE NUMBER	ORIGINATING USER NODE NUMBER	FREEDOM CODE						CUMULATIVE COUNT		
	- X -	- Y -	- Z -			1	2	3	4	5	6	--F--	--R--	--S--
20	579.989	866.032	0.000	1	20	F	F	F	F	F	F	6	0	0
14	116.600	150.000	0.000	1	14	R	R	R	R	R	R	6	5	1
3	197.733	100.000	0.000	1	3	R	R	R	R	R	R	6	11	1
2	178.867	50.000	0.000	1	2	F	F	F	F	F	F	12	11	1
13	77.733	100.000	0.000	1	13	F	F	F	F	F	F	18	11	1
1	160.000	0.000	0.000	1	1	S	S	S	S	S	S	18	11	7
101	160.000	0.000	-45.000	1	101	S	S	S	S	S	S	19	11	13
12	38.867	50.000	0.000	1	12	F	F	F	F	F	F	24	11	13
11	0.000	0.000	0.000	1	11	S	S	S	S	S	S	24	11	19
111	0.000	0.000	-45.000	1	111	S	S	S	S	S	S	24	11	25

NOTES:

1. Output from PRINT INPUT (INTERACT, BC) statement. Line 12 of Control Program (sec. 8.6.3).

*** INTERACT GEOMETRIC RETAINS ***

SUBSTRUCTURE NUMBER 1
 PROBLEM ID - EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

DATE 77/04/20 PAGE 1

NODAL COORDINATES		ORIGINATING SUBSTRUCTURE NUMBER	ORIGINATING USER NODE NUMBER	FREEDOM CODE					
- X -	- Y -	- Z -		1	2	3	4	5	6
116.600	150.000	0.000	1	R	R	R	R	R	R
			2	R	R	R	R	R	R
197.733	100.000	0.000	1	R	R	R	R	R	R
			2	R	R	R	R	R	R

*** RETAINED FREEDOM VECTOR ***

SUBSTRUCTURE NUMBER 1
 PROBLEM ID - EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

DATE 77/04/20 PAGE 1

RETAINED FREEDOM VECTOR ROW	USER NODE NUMBER	FREEDOM NUMBER
1	14	1
2	14	2
3	14	3
4	14	4
5	14	5
6	3	1
7	3	2
8	3	3
9	3	4
10	3	5
11	3	6

NOTES:
 1. Output from PRINT INPUT (INTERACT, RETA) statement. Line 12 of Control Program (sec. 8.6.3).

*** INTERACT CONNECTIVITY DATA ***
 SUBSTRUCTURE NUMBER 2
 PROBLEM ID - EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS
 DATE 77/04/20 PAGE 1

NODAL COORDINATES		ORIGINATING SUBSTRUCTURE NUMBER	ORIGINATING USER NODE NUMBER
- X -	- Y -	- Z -	
116.600	150.000	0.000	14
		2	14
197.733	100.000	0.000	3
		2	3

*** INTERACT LOADCASE DATA ***
 SUBSTRUCTURE NUMBER 1
 PROBLEM ID - EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS
 DATE 77/04/20 PAGE 1

LOADCASE IDENTIFIER	ORIGINATING SUBSTRUCTURE NUMBER
AIRLOAD	1
THERMAL	1

NOTES:
 1. Output from PRINT INPUT (INTERACT, CONN, LOAD) statement. Line 12 of Control Program (sec. 8.6.3).

INTERNAL NODE NO.	USER NODE NO.	ANALYSIS FRAME	LOADCASE	TX	TY	TZ	RX	RY	RZ
6	1	SPAN	AIRLOAD THERMAL TIPOAD	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000
4	2	SPAN	AIRLOAD THERMAL TIPOAD	-0.00369 -0.016724 -0.000436	-0.002981 -0.006688 -0.82E-05	-1161681 -269957 -0.008259	-0.01220 -0.071189 -0.000887	-0.01992 -0.120999 -0.002699	0.000000 0.000000 0.000000
3	3	SPAN	AIRLOAD THERMAL TIPOAD	-0.00994 -0.0207017 -0.001044	-0.004041 -0.08005 -8.30E-05	-2990050 -1.13446 -0.031371	-0.02204 -0.086926 -0.001687	-0.03726 -0.0241326 -0.005760	-0.000006 -0.0039541 -0.0007853
9	11	SPAN	AIRLOAD THERMAL TIPOAD	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000
8	12	SPAN	AIRLOAD THERMAL TIPOAD	-0.00444 -0.0073171 -0.0000285	-0.002412 -0.007311 -0.000016	-2113852 -1.156644 -0.008925	-0.01532 -0.010691 -0.001287	-0.02084 -0.0071389 -0.0002031	0.000000 0.000000 0.000000
5	13	SPAN	AIRLOAD THERMAL TIPOAD	-0.00994 -0.0164378 -0.0000575	-0.005513 -0.003522 -0.000030	-5094068 -776470 -0.032429	-0.02831 -0.013941 -0.002792	-0.03280 -0.0168653 -0.004078	-0.000655 -0.004286 -0.0000616
2	14	SPAN	AIRLOAD THERMAL TIPOAD	-0.00805 -0.0241875 -0.0000411	-0.005094 -0.0048371 -0.16E-05	-8393019 -2.03769 -0.071953	-0.03751 -0.010064 -0.0003957	-0.03713 -0.0274061 -0.0006333	0.000000 0.000000 0.000000
7	101	SPAN	AIRLOAD THERMAL TIPOAD	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000
10	111	SPAN	AIRLOAD THERMAL TIPOAD	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000	0.000000 0.000000 0.000000

RUN TIME FOR ABOVE OVERLAY .540 CP SECONDS

NOTES:
1. Output from the PRINT OUTPUT (DISP) statement. Line 24 of Control Program (sec. 8.6.3).
2. Nodal displacements are shown for each loadcase.

STRESSES ** PROBLEM ID -- EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

DATA SET NO. 1

COVER NODES= 1 12 11 MATL=M 1 TEMP= 70
1 T(1)U=-250000 T(1)U=0.00000 T(2)U=0.00000 ALPHA=0.00000 BETAU=0.00000 T(1)U=-250000
T(1)U=0.00000 T(2)U=0.00000 ALPHA=0.00000 BETAU=0.00000
LOADCASE SIGMA1U SIGMA2U SIGMA3U SIG-S1U SIG-S2U SIGMA21 SIGMA12 TAU12L SIG-S1L SIG-S2L
AIRLOAD 41.22561 -5930.09 -751.871 0.000000 0.000000 -41.0697 5773.184 732.8028 0.000000 0.000000
THERMAL -17891.7 -7495.42 -14345.7 0.000000 0.000000 15948.70 7180.302 12850.04 0.000000 0.000000
TIPLoad -3.92526 530.8097 67.50866 0.000000 0.000000 3.822570 -523.788 -66.2088 0.000000 0.000000

COVER NODES= 1 2 13 12 MATL=M 1 TEMP= 70
2 T(1)U=-250000 T(1)U=0.00000 T(2)U=0.00000 ALPHA=0.00000 BETAU=0.00000 T(1)U=-250000
T(1)U=0.00000 T(2)U=0.00000 ALPHA=0.00000 BETAU=0.00000
LOADCASE SIGMA1U SIGMA2U SIGMA3U SIG-S1U SIG-S2U SIGMA21 SIGMA12 TAU12L SIG-S1L SIG-S2L
AIRLOAD -4324.32 258.6967 580.5564 0.000000 0.000000 4165.392 -258.989 -553.255 0.000000 0.000000
THERMAL -354.233 -8657.40 35.28748 0.000000 0.000000 730.4187 7705.182 -117.514 0.000000 0.000000
TIPLoad 602.8517 13.35721 -81.9052 0.000000 0.000000 -589.872 -12.8876 80.16142 0.000000 0.000000

COVER NODES= 2 3 14 13 MATL=M 1 TEMP= 70
3 T(1)U=-250000 T(1)U=0.00000 T(2)U=0.00000 ALPHA=0.00000 BETAU=0.00000 T(1)U=-250000
T(1)U=0.00000 T(2)U=0.00000 ALPHA=0.00000 BETAU=0.00000
LOADCASE SIGMA1U SIGMA2U SIGMA3U SIG-S1U SIG-S2U SIGMA21 SIGMA12 TAU12L SIG-S1L SIG-S2L
AIRLOAD -2765.76 272.4139 305.0981 0.000000 0.000000 2685.833 -257.242 -302.536 0.000000 0.000000
THERMAL -62.8638 -395.014 394.3530 0.000000 0.000000 356.2042 335.5202 -404.674 0.000000 0.000000
TIPLoad 614.4103 1.966333 -81.1581 0.000000 0.000000 -606.013 -2.83101 80.97146 0.000000 0.000000

SPAR NODES= 1 2 MATL=M 1 TEMP= 70
10 T-WEB=-250000 FAREA1U=1.50000 FAREA2U=1.50000 FAREA2L=1.50000 D(1)U=0.00000
D(1)U=0.00000 D(2)U=0.00000 A-LMP1U=1.25000 A-LMP1L=1.14583 A-LMP2U=1.14583
A-LMP2L=1.14583
LOADCASE P-CAPU SIGMA-U P-LMPU P-CAPL SIGMA-L P-LMPL Q-EQUIV TAU-MAX
AIRLOAD -7976.27 -2834.32 -3724.79 7791.733 2765.892 3642.895 644.1459 2810.819
THERMAL 7609.197 -9080.30 21229.64 -6466.94 8210.764 -18783.1 65.82426 287.2332
TIPLoad 1079.304 397.0422 483.7404 -1055.00 -388.032 -472.956 -16.5046 -72.0201

SPAR NODES= 2 3 MATL=M 1 TEMP= 70
11 T-WEB=-250000 FAREA1U=1.50000 FAREA2U=1.50000 FAREA2L=1.50000 D(1)U=0.00000
D(1)U=0.00000 D(2)U=0.00000 A-LMP1U=1.14583 A-LMP1L=1.14583 A-LMP2U=1.14583
A-LMP2L=1.04167
LOADCASE P-CAPU SIGMA-U P-LMPU P-CAPL SIGMA-L P-LMPL Q-EQUIV TAU-MAX
AIRLOAD -6435.83 -2394.75 -2843.70 6086.074 2259.849 2696.300 422.8658 1860.609
THERMAL 6903.016 -8635.60 19856.42 -5623.18 7901.289 -17475.1 78.98912 347.5521
TIPLoad 1138.463 435.0027 485.9590 -1101.87 -420.887 -470.535 -19.8055 -87.1443

NOTES:

1. Output from the PRINT OUTPUT (STRESS) statement. Line 25 of Control Program (sec. 8.6.3).
2. Element stresses are shown for each loadcase.

***** RESULTANT ABOUT THE GLOBAL ORIGIN OF LOADS APPLIED TO SET 1 STAGE 1 *****

LOADCASE	F1	F2	F3	M1	M2	M3
AIRLOAD	0.	0.	.50016E+05	.27842E+07	-.54019E+07	0.
THERMAL	-.26193E-08	-.19558E-07	-.24011E-09	.32037E-06	.63330E-07	-.13751E-05

LOADS (FREE PARTITION)-PROB 10 -EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS DATE 77/04/20 PAGE 1

SUBSTRUCTURE NUMBER 1

INTERNAL NODE NO.	USER NODE NO.	ANALYSIS FRAME	LOADCASE	1	2	3	4	5	6
4	2	SPAN	AIRLOAD	0.000000	0.000000	8752.734	0.000000	0.000000	0.000000
			THERMAL	9024.576	-54843.9	8492.991	-.131E+08	2142548.	0.000000
5	13	SPAN	AIRLOAD	0.000000	0.000000	9211.210	0.000000	0.000000	0.000000
			THERMAL	8974.607	65235.43	-17638.6	-.14E+08	1942703.	0.000000
8	12	SPAN	AIRLOAD	0.000000	0.000000	10169.84	0.000000	0.000000	0.000000
			THERMAL	8974.607	65235.43	-17640.3	-.16E+08	2136040.	0.000000

NOTES:

1. Output from PRINT OUTPUT (LOADS) statement. Line 26 of Control Program (sec. 8.6.3).

a) Resultants of the applied loads for 6 degrees of freedom and for each loadcase.

b) The cumulated equivalent nodal loads as applied to the 6 degrees of freedom of each node for each loadcase. These data are printed in three blocks corresponding to the FREE, RETAIN, and SUPPORT nodal freedoms. An example is shown on this page.

DATA SET NO. 1, STAGE NO. 1

INTERNAL NODE NO.	USER NODE NO.	ANALYSIS FRAME	LOADCASE	1	2	3	4	5	6
13	1	SPAN	AIRLOAD TIPOLOAD THERMAL	2590.367 -260.883 -22704.3	-33.4512 -15.0879 22986.40	-32241.7 998.5022 -543.710	-47958.7 17444.47 -58E+07	3164752. -367254. -542E+07	0.000000 0.000000 0.000000
14	101	SPAN	AIRLOAD TIPOLOAD THERMAL	-2348.45 246.2522 8616.579	-385.640 40.43746 1414.909	-1002.22 105.0604 3675.732	1.680454 -286470 -18.4594	21.09995 -1.18085 -29.3949	3.472125 -327768 -14.9908
16	11	SPAN	AIRLOAD TIPOLOAD THERMAL	2819.145 -180.355 9651.593	-3.49676 1.569367 -23789.0	-35179.5 826.5034 -4722.60	136642.5 -3701.85 6237270.	4007395. -297419. 5086670.	0.000000 0.000000 0.000000
17	111	SPAN	AIRLOAD TIPOLOAD THERMAL	-3061.06 194.9860 4436.088	422.5878 -26.9189 -612.277	-1098.25 69.93406 1590.578	8.313471 -510166 24.62665	32.05427 -1.05602 -10.3771	2.166987 -204214 14.06652

NOTES:

1. Output from PRINT OUTPUT (REACTIONS) statements. Line 27 of Control Program (sec. 8.6.3).
2. Reactions are printed for 6 degrees of freedom of each node for each loadcase. For a substructured analysis, these data are printed in two blocks corresponding to the SUPPORT and RETAIN nodal freedoms, where the RETAIN reactions correspond to the substructure interacting freedoms. Examples are shown on this page and the next two pages.

REACTIONS(RETAIN PARTITION)-PROB ID -EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

DATE 77/04/20 PAGE 1

SUBSTRUCTURE NUMBER 1

INTERNAL NODE NO.	USER NODE NO.	ANALYSIS FRAME	FREEDOM NUMBER	LOADCASE
2	14	SPAN	1	AIRLOAD THERMAL TIPLCAD -4.47559 16.40533 -4720981
2	14	SPAN	2	AIRLOAD THERMAL TIPLCAD -155.844 571.2479 16.43887
2	14	SPAN	3	AIRLOAD THERMAL TIPLCAD 8664.064 3039.707 -929.423
2	14	SPAN	4	AIRLOAD THERMAL TIPLCAD 84432.86 -127981. -3912.70
2	14	SPAN	5	AIRLOAD THERMAL TIPLCAD -334718. -785386. 162971.5
3	3	SPAN	1	AIRLOAD THERMAL TIPLCAD 4.475594 -16.4553 -472098
3	3	SPAN	2	AIRLOAD THERMAL TIPLCAD 155.8441 -571.248 -16.4389
3	3	SPAN	3	AIRLOAD THERMAL TIPLCAD 10842.03 -3039.71 -1070.58
3	3	SPAN	4	AIRLOAD THERMAL TIPLCAD 44574.53 -161592. -11471.1
3	3	SPAN	5	AIRLOAD THERMAL TIPLCAD -986797. 793702.4 236363.9
3	3	SPAN	6	AIRLOAD THERMAL TIPLCAD 0.000000 -23E-12 0.000000

REACTIONS(RETAIN PARTITION)-PROB 10 -EXAMPLE OF SUBSTRUCTURED STRESS ANALYSIS

SUBSTRUCTURE NUMBER 2

INTERNAL NODE NO.	USER NODE NO.	ANALYSIS FRAME	FREEDOM NUMBER	LOADCASE
9	14	SPAN	1	AIRLOAD THERMAL TIPLCAD 4.475594 -16.4053 -.472098
9	14	SPAN	2	AIRLOAD THERMAL TIPLCAD 155.8441 -571.248 -16.4389
9	14	SPAN	3	AIRLOAD THERMAL TIPLCAD -8664.06 -3039.71 929.4226
9	14	SPAN	4	AIRLOAD THERMAL TIPLCAD -84432.9 127981.2 3912.696
9	14	SPAN	5	AIRLOAD THERMAL TIPLCAD 334718.5 785386.3 -162972.
10	3	SPAN	1	AIRLOAD THERMAL TIPLCAD -4.47559 16.40533 .4720981
10	3	SPAN	2	AIRLOAD THERMAL TIPLCAD -155.844 571.2479 16.43887
10	3	SPAN	3	AIRLOAD THERMAL TIPLCAD -10842.0 3039.707 1070.577
10	3	SPAN	4	AIRLOAD THERMAL TIPLCAD -44574.5 161591.8 11471.07
10	3	SPAN	5	AIRLOAD THERMAL TIPLCAD 986796.7 -793702. -236364.
10	3	SPAN	6	AIRLOAD THERMAL 0.000000 0.000000

8.7 EXAMPLE 7: SUBSTRUCTURED VIBRATION ANALYSIS

8.7.1 Description of Model

The total structural model used in this example is the box beam described in section 8.0. It is analyzed using two substructures as illustrated previously in figure 8-7.

8.7.2 Description of Analysis

This analysis is similar to that of EXAMPLE 4; the only difference being that the structural model is defined as two separate substructures.

8.7.3 Execution-Control Data

```
1 BEGIN CONTROL PROGRAM SUBVIB
2 PROBLEM ID (EXAMPLE SUBSTRUCTURED VIBRATION ANALYSIS)
3 USER COMMON (I)
4 READ INPUT (MODE2)
5 PRINT INPUT (INTERACT,SS=(1,2),NODE,CONN,RETA,BC)
6 DO 20 I=1,2
7 EXECUTE STIFFNESS (SET=I)
8 EXECUTE MASS (SET=I,OPTION=3)
9 20 CONTINUE
10 PERFORM SS-MERGE (STIF,SS=(1,2))
11 PERFORM SS-REDU (STIF,SS=(1,2))
12 PERFORM SS-MERGE (STIF,SS=3)
13 PERFORM SS-MERGE (MASS,SS=3)
14 PERFORM SS-VSOL (SS=3)
15 EXECUTE VIBRATION (MASS=MRED003,STIF=KRED003,NFREQS=8)
16 PRINT OUTPUT (VIBRATION)
17 EXECUTE EXTRACT (EXNAME=VIBR,LSUB=VMODE,VSET=1,MODE=(1 TO 8),
1 BSUB=ON001)
18 EXECUTE GRAPHICS (GNAME=VIBR,TYPE=ORTH,OFFLINE=SC4020,
1 VECTOR3=VMODE,VSCALE=50.0,EXNAME=VIBR)
19 END CONTROL PROGRAM
```

NOTES:

- ① Required first statement in an ATLAS execution control deck
- ② Assigns a title to the problem. This title will appear in the printout
- ③ Defines the FORTRAN variable I to be stored in a common block so that the value assigned to I by this Control Program is not destroyed by subsequent ATLAS statements

- ④ Reads and checks the model-definition data
- ⑤ Requests formatted printout of the substructure INTERACT data for substructures 1 and 2
- ⑥-⑨ Execute the STIFFNESS and MASS processors to obtain element stiffness matrices and nodal mass matrices for substructures 1 and 2
- ⑩, ⑪ Transform the element stiffnesses into nodal stiffnesses for substructures 1 and 2
- ⑫ Combines the nodal stiffnesses for substructures 1 and 2 to form a gross stiffness matrix for substructure 3
- ⑬ Combines the nodal masses for substructures 1 and 2 to form a gross mass matrix for substructure 3
- ⑭ Reduces the nodal mass and stiffness matrices. In this particular example, no reduction is involved because all the interacting freedoms are retained; the net result is that the gross mass and stiffness matrices are assigned the names MRED003 and KRED003, respectively.
- ⑮ Calculates the eight lowest-frequency vibration modes
- ⑯ Prints the vibration modes
- ⑰ Extracts the vibration modes and geometry data for graphical display
- ⑱ Generates SC4020 plots of the vibration modes
- ⑲ Required last statement in an ATLAS execution control deck

8.7.4 Model-Definition Data

```

1 BEGIN NODAL DATA
2 REC SPAN 80. 0. 0., 253.2 300. 0., 80. 0. 100.
3 20 1000. 0. 0.
4 RESUME GLOBAL
5 ANALYSIS FRAME SPAN
6 1 160. 0. 0. 15. TO 3 197.73333 100. 0. 12.5
7 11 0. *4 14 116.6 150. 0. 11.25
8 101 160. 0. -45.
9 111 0. *2
10 REORDER FROM 20
11 SET 2
12 REC SPAN 80. 0. 0., 253.2 300. 0., 80. 0. 100.

```

```

13      21 1001. 0. 0.
14      RESUME GLOBAL
15      ANALYSIS FRAME SPAN
16      3 197.73333 100. 0. 12.5 TO 7 273.2 300. 0. 7.5
17      14 116.6 150.0 0. 11.25 TO 17 233.2 300. 0. 7.5
18      REORDER FROM 21
19      END NODAL DATA

20      BEGIN STIFFNESS DATA
21          BEGIN ELEMENT DATA
22              SPAR N10 1 2 .25 1.5
23              SPAR N11 2 3 .25 1.5
24              SPAR N20 11 12 *2 TO N22 13 14
25              COVER N1 1 12 11 .25
26              COVER N2 1 2 13 12 .25 TO N3 2 3 14 13
27              BEAM N51 13 111 .25 0. 0. .06 .03 .03
28              BEAM N52 3 101 **
29          END ELEMENT DATA
30      SET 2
31          BEGIN ELEMENT DATA
32              SPAR N12 3 4 .25 1.5 TO N15 6 7
33              SPAR N23 14 15 .25 1.5 TO N25 16 17
34              SPAR N30 7 17 *2
35              COVER N4 3 4 15 14 .25 TO N6 5 6 17 16
36              COVER N7 6 7 17 .25
37          END ELEMENT DATA
38      END STIFFNESS DATA

39      BEGIN BC DATA
40          SET 1 STAGE 1
41              RETAIN TX TY TZ RX RY FOR 1 TO 100
42              FREE ALL FOR 20
43              SUPPORT ALL FOR 1 11 101 111
44          SET 2 STAGE 1
45              RETAIN TX TY TZ RX RY FOR 1 TO 100
46              FREE ALL FOR 21
47      END BC DATA

48      BEGIN INTERACT DATA
49      UNIQUE NODE NUMBERS
50          DEFINE SS 1 AS SET 1 STAGE 1
51          DEFINE SS 2 AS SET 2 STAGE 1
52          SS 3
53          INTERACT 1 2
54          NO GEOMETRIC RETAINS
55      BEGIN BC CHANGES
56          SS 3
57          REFERENCE SS 1
58          RETAIN TX TY TZ RX RY FOR 3 14
59      END BC CHANGES

```

```

60      DEFINE HIGHEST SS 3 AS SET 3
61      END INTERACT DATA

62      BEGIN SUBSET DEFINITION
63          SUBSETS OF NODAL SET 3
64              ON001 = 2 12 13 3 4 14 15 5 6 16 17 7 +
64                  6 5 4 3 2 12 13 14 15 16 17
65      END SUBSET DEFINITION

66      BEGIN MASS DATA
67          BEGIN CONDITION DATA
68              STAGE 1 CONDITION 1 0 0 0
69          END CONDITION DATA
70      END MASS DATA
71      BEGIN MASS DATA
72          SET 2
73          BEGIN CONDITION DATA
74              STAGE 1 CONDITION 1 0 0 0
75          END CONDITION DATA
76      END MASS DATA

77      END PROBLEM DATA

```

NOTES:

- ①④⑦ Define the NODAL, STIFFNESS, and BOUNDARY CONDITION data for SET numbers 1 (defaulted) and 2; similar data records are described for the EXAMPLE 4 problem where the same nodes, elements, and boundary conditions are defined for the non-substructured model
- ④⑧ Identifies the start of the INTERACT data set
- ④⑨ Specifies that unique node numbers have been used throughout the substructured model. Nodes 3 and 14 occur in both substructures.
- ⑤① Specifies that substructure 1 corresponds to SET 1 and STAGE 1
- ⑤② Defines substructure 2
- ⑤③ Introduces substructure 3
- ⑤④ Defines substructure 3 as the union of substructures 1 and 2
- ⑤⑤ Specifies that only the retained freedoms (TX, TY, TZ, RX and RY) for the interacting nodes (nodes 3 and 14) are

to be interacted. If this record were omitted, freedom RZ would be included in the interaction.

- (55) Indicates that the boundary conditions for the model are to be modified after accounting for any interaction
- (56) Indicates that boundary conditions for substructure 3 are to be changed
- (57) Indicates that the nodes are identified using the numbers assigned in nodal data SET 1. This record is redundant since the UNIQUE NODE NUMBERS option has been specified.
- (58) Specifies the freedoms that are to be retained for the interacting nodes. If this record were omitted, nodes 3 and 14 would not be retained in substructure 3.
- (59) Identifies the end of the BC change data subset
- (60) Defines substructure 3 as the highest level substructure in the interact diagram and assigns to it SET number 3 and STAGE number 1
- (61) Identifies the end of the INTERACT data set
- (62-65) Define an ordered nodal subset to permit vibration modes to be plotted in conjunction with a grid of lines to enhance visibility. Note that for a higher level substructure, only retained nodes may be included in a subset, so nodes 1 and 11 have been omitted.
- (66-76) Define the MASS data for SET numbers 1 and 2. Similar data records are described for the EXAMPLE 4 problem where the same masses are described for the non-substructured model.
- (70,71) These records are required. The MASS preprocessor cannot process more than one substructure in a MASS data set.
- (77) Identifies the end of the model-definition data deck

8.7.5 Results

Representative output from the substructured vibration analysis is shown on the following pages.

*** BOUNDARY CONDITION DATA ***

PAGE 1

DATE 78/04/11

1

SUBSTRUCTURE NUMBER

PROBLEM 10 - EXAMPLE SUBSTRUCTURED VIBRATION ANALYSIS

FREEDOM ACTIVITIES ARE DENIED BY THE FOLLOWING

CHARACTER F DENOTES A PARTITION 1 FREEDOM (F=FREE)
 CHARACTER R DENOTES A PARTITION 2 FREEDOM (R=RETAIN)
 CHARACTER S DENOTES A PARTITION 3 FREEDOM (S=SUPPORT)
 A BLANK DENOTES A FREEDOM WITH ZERO STIFFNESS

USER NODE NUMBER	NODAL COORDINATES			ORIGINATING SUBSTRUCTURE NUMBER	ORIGINATING USER NODE NUMBER	FREEDOM CODE						CUMULATIVE COUNT		
	- X -	- Y -	- Z -			1	2	3	4	5	6	--F--	--R--	--S--
20	579.989	866.032	0.000	1	20	F	F	F	F	F	F	6	0	0
14	116.609	150.000	0.000	1	14	R	R	R	R	R	F	7	5	0
3	197.733	100.000	0.000	1	3	R	R	R	R	R	F	8	10	0
2	178.867	50.000	0.000	1	2	R	R	R	R	R	F	9	15	0
13	77.733	100.000	0.000	1	13	R	R	R	R	R	F	10	20	0
1	160.000	0.000	0.000	1	1	S	S	S	S	S	S	10	20	6
101	160.000	0.000	-45.000	1	101	S	S	S	S	S	S	10	20	12
12	38.867	50.000	0.000	1	12	R	R	R	R	R	F	11	25	12
11	0.000	0.000	0.000	1	11	S	S	S	S	S	S	11	25	18
111	0.000	0.000	-45.000	1	111	S	S	S	S	S	S	11	25	24

THE ABOVE BOUNDARY CONDITION DATA DOES NOT REFLECT FREEDOM ACTIVITY

NOTES:

1. Output from PRINT INPUT (INTERACT, BC) statement, line 5 of Control Program (sec. 8.7.3).
2. The input INTERACT data are shown by lines 48-61 in section 8.7.4.
3. Inactive freedoms are not identified by this printout because the INTERACT data were read prior to execution of the STIFFNESS processor.
4. This printout shows the BC data for substructure 1, whereas the BC data for substructure 2 are shown on the next page.

*** BOUNDARY CONDITION DATA ***

FREEDOM ACTIVITIES ARE DENOTED BY THE FOLLOWING

CHARACTER F DENOTES A PARTITION 1 FREEDOM (F=FREE)
 CHARACTER R DENOTES A PARTITION 2 FREEDOM (R=RETAIN)
 CHARACTER S DENOTES A PARTITION 3 FREEDOM (S=SUPPORT)
 A BLANK DENOTES A FREEDOM WITH ZERO STIFFNESS

USER NODE NUMBER	NODAL COORDINATES			ORIGINATING SUBSTRUCTURE NUMBER	ORIGINATING USER NODE NUMBER	FREEDOM CODE						CUMULATIVE COUNT		
	- X -	- Y -	- Z -			1	2	3	4	5	6	--F--	--R--	--S--
21	586.489	866.898	0.000	2	21	F	F	F	F	F	F	6	0	0
7	273.200	306.000	0.000	2	7	R	R	R	R	R	R	7	5	0
17	233.200	306.000	0.000	2	17	R	R	R	R	R	R	8	10	0
6	254.333	256.000	0.000	2	6	R	R	R	R	R	R	9	15	0
16	194.333	256.000	0.000	2	16	R	R	R	R	R	R	10	20	0
5	235.467	206.000	0.000	2	5	R	R	R	R	R	R	11	25	0
15	155.467	206.000	0.000	2	15	R	R	R	R	R	R	12	30	0
4	216.600	156.000	0.000	2	4	R	R	R	R	R	R	13	35	0
14	116.600	156.000	0.000	2	14	R	R	R	R	R	R	14	40	0
3	197.733	106.000	0.000	2	3	R	R	R	R	R	R	15	45	0

THE ABOVE BOUNDARY CONDITION DATA DOES NOT REFLECT FREEDOM ACTIVITY

NOTES:
 1. Reference the notes on last page.

THE EQUILIBRIUM OR COMPATIBILITY CHECK GIVEN BELOW IS THE
BIASED ROOT MEAN SQUARE (RMS) WHICH IS A MEASURE OF THE
DEVIATION FROM ZERO OF THE I-TH RESIDUE VECTOR.

TABLE OF EQUILIBRIUM RESULTS

MODE	EIGENVALUE (CUTOFF= .7E-04)	RMS ERROR
1	.7180699E+04	.2214467E-06
2	.2081499E+05	.4325896E-06
3	.6829651E+05	.5040938E-06
4	.7323175E+05	.3030127E-06
5	.1316052E+06	.3886464E-06
6	.1975317E+06	.6204085E-06
7	.3120153E+06	.4599367E-06
8	.3517515E+06	.5989063E-06

NOTES:

1. Printout generated automatically during execution of the VIBRATION processor, line 15 of Control Program (sec. 8.7.3)
2. The eigenvalues are identical to those calculated for EXAMPLE 4 (sec. 8.4); the RMS error is different because of the rounding differences occurring in the two analyses.

THE MODE SHAPE ORTHOGONALITY CHECK, GIVEN BELOW, IS A SYMBOLIC REPRESENTATION OF THE SUB-DIAGONAL LOWER
TRIANGULAR PORTION OF PHI-T X MASS X PHI, WHERE PHI IS THE MATRIX OF MODE SHAPES.

ROW	SYMBOL	INDICATES
0		TERM) 1.E-12
1		1.E-12) TERM) 1.E-08
2		1.E-08) TERM) 1.E-04
X		1.E-04) TERM

0
2 0
3 00
4 000
5 0000
6 00000
7 000000
8 0000000

RUN TIME FOR ABOVE OVERLAY 1.273 CP. SECONDS

FREQUENCIES WITH INDEX NAME = FREQS01

MODE	HERTZ
1	.1348663E+02
2	.2296192E+02
3	.4159292E+02
4	.6306950E+02
5	.5773733E+02
6	.7073568E+02
7	.8890130E+02
8	.9439263E+02

W A P N I N G -- FREEDOM AND NODE NUMBERS NOT PRINTED BECAUSE ROW SIZE OF RETAINED FREEDOM VECTOR (25)
 DOES NOT EQUAL ROW SIZE OF MODE SHAPE MATRIX (60). CHECK SET AND STAGE NUMBERS.

NOTES:

1. Output from the PRINT OUTPUT (VIBRATION) statement, line 16 of Control Program (sec. 8.7.3).
2. The number of frequencies printed is specified by the key-word NFREQS in the EXECUTE VIBRATION statement; line 15 of the Control Program.
3. The WARNING message is commonly encountered as a result of not specifying the SET and STAGE numbers in the EXECUTE VIBRATION statement.

SOLUTION MODE SHAPES
WITH INDEX NAME = MODES01

KJ4	MODE 1	MODE 2	MODE 3	MODE 4	MODE 5	MODE 6	MODE 7	MODE 8	
1	-.00014391	-.00093896	-.00035224	-.04152713	-.00022392	-.00032869	-.00043928	-.00067473	1
2	-.00011610	-.00102504	-.00212944	-.07046550	-.00085772	-.00036151	-.00020686	-.00071142	2
3	-.03240073	-.15188939	-.18188038	-.00536352	-.45595850	-.17055563	-.62495349	-.20181451	3
4	-.00071588	-.00366688	-.00094170	-.00001635	-.00645134	-.00711763	-.00212285	-.01498103	4
5	-.00096032	-.00259241	-.00206310	-.00002483	-.00065414	-.00011409	-.00467019	-.00401026	5
6	-.00017246	-.00040472	-.00141065	-.02912868	-.00057177	-.00042509	-.00028094	-.00001420	6
7	-.0009380	-.00020055	-.00081308	-.01790051	-.00038767	-.00023684	-.00020700	-.00012401	7
8	-.06234390	-.13384257	-.47961759	-.00311832	-.21746080	-.84500546	-.08440241	-.19811421	8
9	-.00070481	-.0011646	-.00281529	-.00002621	-.0026415	-.00288766	-.00148704	-.00774269	9
10	-.00101491	-.0013803	-.00165702	-.00002003	-.00045839	-.00208233	-.00015820	-.00036152	10
11	-.00038597	-.00130221	-.00354722	-.08215587	-.0013064	-.00091525	-.00050308	-.00006109	11
12	-.00021459	-.00085837	-.00239247	-.06373141	-.00093578	-.00054421	-.00037094	-.00030244	12
13	-.19636037	-.36606148	-.84469404	-.00601807	-.26583214	-.54934622	-.07391429	-.34265971	13
14	-.00156095	-.00364819	-.00311990	-.00004139	-.00578460	-.00493970	-.00161843	-.01313008	14
15	-.00191670	-.00244452	-.00076660	-.00001604	-.00079198	-.00700201	-.00105701	-.00110669	15
16	-.00038442	-.00156158	-.00043689	-.09305568	-.00015292	-.00120015	-.00081786	-.0003082	16
17	-.00014172	-.00457200	-.00822753	-.33370761	-.00381574	-.00011967	-.00022753	-.00051349	17
18	-.26097304	-.75569733	-.63141508	-.00812479	-.50423117	-.31726834	-.18759364	-.84220373	18
19	-.00184470	-.01176018	-.00656640	-.00006052	-.00093342	-.01290842	-.01376093	-.03593787	19
20	-.00328858	-.00234680	-.00068890	-.00006704	-.000675052	-.00725054	-.00090555	-.00035348	20
21	-.00037800	-.00131991	-.00038177	-.09170971	-.00043284	-.00134476	-.00077171	-.00008713	21
22	-.00010040	-.00667506	-.01246413	-.53205024	-.00677808	-.00099843	-.00038376	-.00096201	22
23	-.46671598	-.97155060	-.50442162	-.00105842	-.04640293	-.77120001	-.16634482	-.66416296	23
24	-.00164014	-.01084104	-.01084104	-.00006593	-.00589866	-.00472865	-.01292413	-.05085427	24
25	-.00435740	-.0019533	-.00779258	-.00012855	-.00784084	-.00181721	-.00078590	-.00175359	25
26	-.00037176	-.00101290	-.0009029	-.07042610	-.00021908	-.00132639	-.0007642	-.00307642	26
27	-.00060095	-.00885717	-.01734550	-.76110796	-.01055937	-.00217500	-.00066847	-.00338152	27
28	-.72747798	1.00000000	-.09628248	-.00949573	-.64717668	-.54117305	-.42782088	-.46502941	28
29	-.00039618	-.00434353	-.00754197	-.00004348	-.00459201	-.00306311	-.01151914	-.05234316	29
30	-.00491598	-.00200617	-.01447683	-.00017351	-.00643695	-.01049784	-.00937177	-.00406621	30
31	-.00036534	-.00059223	-.00085650	-.03469141	-.00035938	-.00115033	-.00083682	-.00044213	31
32	-.0002116	-.01108986	-.02242562	1.00000000	-.01460678	-.00364612	-.0099177	-.00516232	32
33	1.00000000	-.86162346	1.00000000	-.02039336	1.00000000	-.44270063	1.00000000	-.34970943	33
34	-.00245148	-.01217756	-.00709686	-.00000193	-.00875931	-.02086804	-.01665126	-.03831552	34
35	-.00466285	-.00030329	-.01410358	-.00018570	-.00728207	-.01211336	-.01004172	-.00217313	35
36	-.00032552	-.00135667	-.00467263	-.14539925	-.00285723	-.00032054	-.00085915	-.00160585	36
37	-.00012983	-.00507200	-.00915106	-.37682589	-.00440574	-.00027441	-.0026272	-.00027230	37
38	-.61123520	-.67083979	-.15900840	-.00849949	-.59680186	-.36236158	-.35468046	-.52829719	38
39	-.00283511	-.01470425	-.01053640	-.00004464	-.00039250	-.01221120	-.01526061	-.04270424	39
40	-.00266400	-.00394496	-.00796939	-.00008802	-.00201079	-.00712281	-.00289567	-.00303886	40
41	-.00033237	-.00111054	-.00464824	-.14375434	-.00320683	-.00023666	-.00100051	-.00218018	41
42	-.00008365	-.00760687	-.01449110	-.62684010	-.00830948	-.00145949	-.00050126	-.00189538	42
43	-.82773965	-.32924528	-.59425015	-.00159813	-.54458521	-.60664164	-.69927780	-.34038275	43
44	-.00192327	-.01234055	-.01441413	-.00004283	-.00743202	-.00176610	-.0027780	-.05658376	44
45	-.00233780	-.00972613	-.00573651	-.00017423	-.00845324	-.00722866	-.01314494	-.00588010	45
46	-.00033915	-.00078710	-.00410013	-.11789125	-.00290248	-.00000000	-.00000000	-.00588010	46
47	-.00000000	-.00000000	-.02048480	-.90866287	-.01303316	-.00000000	-.00000000	-.00588010	47
48			-.9521856	-.01651268	-.4840725			-.00588010	48
49			-.0001110	-.0001110	-.0026197			-.00588010	49
50			-.00020429	-.00020429	-.0154			-.00588010	50
51			12286572					-.00588010	51

NOTES:

- Output from PRINT OUTPUT (VIBRATION) statement. Line 16 of Control Program (sec. 8.7.3).
- These modes are equivalent to those calculated for EXAMPLE 4 (sec. 8.4), but the printout order is different. Optionally, the order can be specified by the INTERACT data set.

GENERALIZED MASS
WITH INDEX NAME = GNASS01

MODE	1	2	3	4	5	6	7	8
1	.1102234E+01	-.6039613E-13	.4751755E-13	-.2775558E-15	-.2614575E-13	.4329870E-14	.7771561E-15	.9103829E-14
2	-.6039613E-13	.2345088E+01	.4085621E-13	-.1097039E-13	-.8104628E-14	.8437695E-14	-.5329071E-14	.1021405E-13
3	.4751755E-13	.4085621E-13	.2107121E+01	-.1402186E-12	-.7993606E-13	-.2842171E-13	-.1421085E-13	-.7105427E-14
4	-.2775558E-15	-.1097039E-13	-.1402186E-12	.8837760E+00	.1665335E-14	.2206568E-14	-.1845746E-14	-.1637579E-14
5	-.2614575E-13	-.8104628E-14	-.7993606E-13	.1665335E-14	.1514362E+01	.2708944E-13	-.1376677E-13	-.1465494E-13
6	.4329870E-14	.8437695E-14	-.2842171E-13	.2206568E-14	.2708944E-13	.2188789E+01	-.3330669E-15	-.1654232E-13
7	.7771561E-15	-.5329071E-14	-.1421085E-13	-.1845746E-14	-.1376677E-13	-.3330669E-15	.1094208E+01	.8881784E-15
8	.9103829E-14	-.1021405E-13	-.7105427E-14	-.1637579E-14	-.1465494E-13	-.1654232E-13	.8881784E-15	.1767473E+01

GENERALIZED STIFFNESS
WITH INDEX NAME = GSTIF01

MODE	1	2	3	4	5	6	7	8
1	.7914811E+04	0.	0.	0.	0.	0.	0.	0.
2	0.	.4881299E+05	0.	0.	0.	0.	0.	0.
3	0.	0.	.1439090E+06	0.	0.	0.	0.	0.
4	0.	0.	0.	.6472046E+05	0.	0.	0.	0.
5	0.	0.	0.	0.	.1992979E+06	0.	0.	0.
6	0.	0.	0.	0.	0.	.4323553E+06	0.	0.
7	0.	0.	0.	0.	0.	0.	.3414096E+06	0.
8	0.	0.	0.	0.	0.	0.	0.	.6217113E+06

RUN TIME FOR ABOVE OVERLAY .115 CP SECONDS

NOTES:

1. Output from PRINT OUTPUT (VIBRATION) statement. Line 16 of Control Program (sec. 8.7.3).
2. The off-diagonal terms of the generalized mass matrix are a measure of the rounding error. Therefore, they are different from those shown for EXAMPLE 4 (sec. 8.4).

8.8 EXAMPLE 8: STRUCTURAL DESIGN

8.8.1 Description of Analysis

This example consists of a fully stressed design of the wing box described in section 8.0. The box is fully supported at the root. Three loading conditions are applied:

- a) A distributed pressure load acting in the GLOBAL Z direction. The pressure is uniform over each element and varies from a maximum at the root to a minimum at the tip.
- b) A concentrated tip load acting in the GLOBAL Z direction and divided equally between front and rear spars.
- c) A thermal loading consisting of a uniform ΔT of 200°F on the upper surface and -200°F on the lower surface.

The ratio of section modulus to area for the BEAMS is taken to be 0.5. The following lower bounds are specified:

- Front and rear spar web thickness of 0.03, flange areas of 1 and stiffener areas of 0.
- Tip rib web thickness of 0.036, flange areas of 1.1 and stiffener areas of 0.

The following elements are sized to a specified margin:

- Rear spar to +0.05
- Cover elements 2 through 6 to +0.06

The following material property codes (ATLAS standard materials) are used for calculating margins of safety:

- M1 for the SPAR elements
- M2 for the COVER elements
- M3 for the BEAM elements

Note that the BEAM elements are not resized.

Resize cycles are performed until one of the following conditions is satisfied:

- a) Five cycles have been performed
- b) The weight change between two consecutive cycles is less than 100
- c) The ratio of weight change to new total weight for two consecutive cycles is less than 7 percent

8.8.2 Execution-Control Data

```
1 BEGIN CONTROL MATRIX PROGRAM UG04
2 PROBLEM ID (FULLY STRESSED DESIGN EXAMPLE)
3 USER COMMON (BEGCYCL,ENDCYCL,CURCYCL,CONVERG,ISET,
4 1 ISTAGE,IPOS)
5 INTEGER BEGCYCL,ENDCYCL,CURCYCL
6 DIMENSION CONVERG(17)
7 BEGCYCL= 1
8 ENDCYCL= 5
9 CONVERG(14) = 100.
10 CONVERG(15) = .07
11 READ INPUT (MODE2)
12 PRINT INPUT (NODAL)
13 PRINT INPUT (STIFFNESS)
14 PRINT INPUT (LOADS)
15 PRINT INPUT (DESIGN,SIZING)
16 PERFORM DESIGN
17 PRINT INPUT (BC)
18 EXECUTE DESIGN (HISTORY,STAGE=1,CYCLE=1 TO 5,SUB=20)
19 PRINT OUTPUT (DESIGN)
20 PRINT OUTPUT (DESIGN,HISTORY)
21 END CONTROL PROGRAM
```

NOTES:

- ① Required first statement in an ATLAS execution control deck. It should be noted that the word MATRIX is included in this statement. This word is required when the SNARK language is used in a control deck. In this case, the PERFORM DESIGN procedure uses SNARK. The name selected for this Control Program is UG04.
- ② Assigns a title to the problem. This title will appear in the printout.
- ③,④ Defines the FORTRAN variables to be stored in a common block so that their values assigned by this Control

Program are not destroyed between successive ATLAS statements.

- ⑤-⑩ Define values for the FORTRAN variables used to direct and control the automated structural design process; five design cycles are requested along with those design solution convergence criteria described previously.
- ⑪ Directs ATLAS to read and check the model-definition data deck
- ⑫-⑮ Request formatted printout of the following input data:
- ⑫ nodes
 - ⑬ stiffness elements
 - ⑭ loads
 - ⑮ design
- ⑯ Directs ATLAS to perform an automated fully-stressed structural design. The DESIGN execution-control procedure calculates the structural stiffnesses, forms the equilibrium equations, solves for displacements, stresses and reactions, resizes the element properties, and calculates the structural weight variation.
- ⑰ Requests a formatted printout of the boundary condition data
- ⑱ Directs ATLAS to execute the DESIGN processor to extract the resizing history for the elements in subset (region) number 20 defined by the data deck; the search is through the five design CYCLES and for the selected design loadcases.
- ⑲, ⑳ Request printout of the resized elements and the element design history as calculated by the DESIGN processor
- ㉑ Required last statement in an ATLAS execution-control deck

8.8.3 Model-Definition Data

```
1 BEGIN NODAL DATA
2 REC SPAN 80. 0. 0., 253.2 300.0 0., 80. 0. 100.
3 20 1000. 0. 0.
4 RESUME GLOBAL
5 ANALYSIS FRAME SPAN
6 1 160. 0. 0. 15. TO 7 273.2 300. 0. 7.5
7 11 0. 0. 0. 15. TO 17 233.2 300. 0. 7.5
8 101 160. 0. -45.
```

```

9  111  0.  0.  -45.
10 REORDER FROM 20
11 END NODAL DATA

12 BEGIN STIFFNESS DATA
13 BEGIN ELEMENT DATA
14 SPAR N10 1 2 .25 1.5 TO N15 6 7
15 SPAR N20 11 12 .25 1.5 TO N25 16 17
16 SPAR N30 7 17 .25 1.5
17 COVER N1 1 12 11 .25
18 COVER N2 1 2 13 12 .25 TO N6 5 6 17 16
19 COVER N7 6 7 17 .25
20 BEAM N51 13 111 .25 0. 0. .06 .03 .03
21 BEAM N52 3 101 **
22 END ELEMENT DATA
23 END STIFFNESS DATA

24 BEGIN BC DATA
25 SUPPORT ALL FOR 1 11 101 111
26 END BC DATA

27 BEGIN LOADS DATA
28 LOAD CASE ID AIRLOAD *ELEMENT PRESSURE LOADING*
29 LOAD CASE ID TIPLoad *2000 LB. LOAD AT TIP*
30 LOAD CASE ID THERMAL **200 DEG. UPR. AND -200 LWR. #
31 BEGIN NODAL LOAD DATA
32 CASE TIPLoad
33 7 17 FZ -1000.
34 END NODAL LOAD DATA
35 BEGIN ELEMENT LOAD DATA
36 DIRECTION GLOBAL 0. 0. 1.
37 CASE AIRLOAD
38 1 3.5
39 *+6 1 -.5
40 END ELEMENT LOAD DATA
41 BEGIN ELEMENT THERMAL LOAD DATA
42 CASE THERMAL
43 10 TO 15, 20 TO 25, 30 200. -200. 200. -200.
44 1 TO 7 200. -200.
45 END ELEMENT THERMAL LOAD DATA
46 END LOADS DATA

47 BEGIN SUBSET DEFINITION
48 SUBSETS OF STIFFNESS SET 1
49 E1 = 10 TO 15 / REAR SPAR
50 E2 = 20 TO 25 / FRONT SPAR
51 E10 = 2 TO 6 / UPPER AND LOWER SURFACE
52 E11 = BEAMS
53 E20 = ALL
54 END SUBSET DEFINITION

```

```

55 BEGIN DESIGN DATA
56 MODE 1
57 SET 1
58 BEGIN PROPERTY DATA
59   E11 .5
60 END PROPERTY DATA
61 BEGIN LOWER-BOUND DATA
62   SPAR .030 1. *=3 0
63   N30 .036 1.1 *=3 0
64 END LOWER-BOUND DATA
65 BEGIN MARGIN DATA
66   E1 .05
67   E10 .06
68 END MARGIN DATA
69 BEGIN SIZING DATA
70 M1 SPAR
71 M2 COVER
72 M3 E11
73 END SIZING DATA
74 BEGIN RESTRAIN-SIZING DATA
75   E11
76 END RESTRAIN-SIZING DATA
77 END DESIGN DATA
78 END PROBLEM DATA

```

NOTES:

- ①-④⑥ Define the NODAL, STIFFNESS, BOUNDARY CONDITION and LOADS data sets; these data are identical to that shown and described for the EXAMPLE 1 problem.
- ④⑦ Identifies the start of the SUBSET DEFINITION data set
- ④⑧ Specifies that subsets are defined for the structural model designated as SET 1
- ④⑨-⑤③ Define element subsets for:
 - ④⑨ rear spar
 - ⑤① front spar
 - ⑤② upper and lower wing surface
 - ⑤③ all the BEAM elements
 - ⑤④ all the stiffness elements
- ⑤④ Identifies the end of the SUBSET DEFINITION data set
- ⑤⑤ Identifies the start of the DESIGN data set

- (56) Specifies that the input design data are new (MODE1); options are provided for updating the design data between successive design cycles.
- (57) Identifies the design data with the structural model designated as SET 1
- (58-60) Define the PROPERTY data subset which is required when BEAM elements are included in the structural model to convert the "stress calculated" element loads to element stresses
- (61-64) Define the LOWER-BOUND data subset; the minimum gages allowed for the model are described in section 8.8.1
- (65-68) Define the MARGIN data subset; the rear spar (element subset E1) and the skins (element subset E10) are to be sized such that the corresponding property margins of safety are 0.05 and 0.06, respectively.
- (69-73) Define the SIZING data subset; the allowable stresses to be used for calculating margins of safety for the various elements are those associated with the ATLAS standard materials (codes M1, M2 and M3)
- (74-76) Define the RESTRAIN-SIZING data subset; the wing struts identified by the element subset E11 are not to be resized
- (78) Identifies the end of the model-definition data deck

8.8.4 Results

Representative output from the fully stressed design is shown on the following pages. Printout of input data (nodal, stiffness, boundary condition, and loads) and output data (design and design history) are included. A plot of total weight vs. design cycle is presented in figure 8-8.

***** NODAL DATA ***** DATA SET NUMBER = 1
 PROBLEM ID --- FULLY STRESSED DESIGN EXAMPLE

USER NODE	INPUT RECORD	INTERNAL NODE	X COORD	Y COORD	Z COORD	DX COORD	DY COORD	DZ COORD	NODE REF. FRAME	ANALYSIS REF. FRAME
1	6	13	160.0000	0.0000	0.0000	0.0000	0.0000	15.0000	SPAN	SPAN
2	6	11	178.8667	50.0000	0.0000	0.0000	0.0000	13.7500	SPAN	SPAN
3	6	10	197.7333	100.0000	0.0000	0.0000	0.0000	12.5000	SPAN	SPAN
4	6	8	216.6000	150.0000	0.0000	0.0000	0.0000	11.2500	SPAN	SPAN
5	6	6	235.4667	200.0000	0.0000	0.0000	0.0000	10.0000	SPAN	SPAN
6	6	4	254.3333	250.0000	0.0000	0.0000	0.0000	8.7500	SPAN	SPAN
7	6	2	273.2000	300.0000	0.0000	0.0000	0.0000	7.5000	SPAN	SPAN
11	7	16	0.0000	0.0000	0.0000	0.0000	0.0000	15.0000	SPAN	SPAN
12	7	15	38.8667	50.0000	0.0000	0.0000	0.0000	13.7500	SPAN	SPAN
13	7	12	77.7333	100.0000	0.0000	0.0000	0.0000	12.5000	SPAN	SPAN
14	7	9	116.6000	150.0000	0.0000	0.0000	0.0000	11.2500	SPAN	SPAN
15	7	7	155.4667	200.0000	0.0000	0.0000	0.0000	10.0000	SPAN	SPAN
16	7	5	194.3333	250.0000	0.0000	0.0000	0.0000	8.7500	SPAN	SPAN
17	7	3	233.2000	300.0000	0.0000	0.0000	0.0000	7.5000	SPAN	SPAN
20	3	1	578.9890	866.0318	0.0000	0.0000	0.0000	0.0000	SPAN	SPAN
101	8	14	160.0000	0.0000	-45.0000	0.0000	0.0000	0.0000	SPAN	SPAN
111	9	17	0.0000	0.0000	-45.0000	0.0000	0.0000	0.0000	SPAN	SPAN

RUN TIME FOR ABOVE OVERLAY .048 CP SECONDS

NOTES:

1. Output from PRINT INPUT (NODAL) statement. Line 12 of Control Program (sec. 8.8.2).
2. The input nodal data are shown by lines 1-11 in section 8.8.3.
3. Default printout order is by increasing user-assigned node numbers.
4. All coordinates are printed relative to the GLOBAL reference frame.
5. A blank in the "reference frame" columns denotes the GLOBAL reference frame.

*** STIFFNESS ELEMENT DATA *** DATA SET NUMBER = 1
 PROBLEM ID -- FULLY STRESSED DESIGN EXAMPLE

USER ID

1	COVER	INPUT RECORD USER NODES	17, INTERNAL NODES	21,	TEMPERATURE PROPERTIES	70,	MATERIAL CODE	1
		1	13		T(0)U =	-25000		
		12	15		T(1)U =	0.00000		
		11	16		T(2)U =	0.00000		
					ALPHAU =	0.00000		
					BETAU =	0.00000		
					T(0)I =	-25000		
					T(1)I =	0.00000		
					T(2)I =	0.00000		
					ALPHAI =	0.00000		
					BETAI =	0.00000		
2	COVER	INPUT RECORD USER NODES	18, INTERNAL NODES	17,	TEMPERATURE PROPERTIES	70,	MATERIAL CODE	1
		1	13		T(0)U =	-25000		
		2	11		T(1)U =	0.00000		
		13	12		T(2)U =	0.00000		
		12	15		ALPHAU =	0.00000		
					BETAU =	0.00000		
					T(0)I =	-25000		
					T(1)I =	0.00000		
					T(2)I =	0.00000		
					ALPHAI =	0.00000		
					BETAI =	0.00000		
3	COVER	INPUT RECORD USER NODES	18, INTERNAL NODES	14,	TEMPERATURE PROPERTIES	70,	MATERIAL CODE	1
		2	11		T(0)U =	-25000		
		3	10		T(1)U =	0.00000		
		14	9		T(2)U =	0.00000		
		13	12		ALPHAU =	0.00000		
					BETAU =	0.00000		
					T(0)I =	-25000		
					T(1)I =	0.00000		
					T(2)I =	0.00000		
					ALPHAI =	0.00000		
					BETAI =	0.00000		

NOTES:

1. Output from PRINT INPUT (STIFFNESS) statement. Line 13 of Control Program (sec. 8.8.2).
2. The input stiffness data are shown by lines 12-23 in section 8.8.3.
3. Default printout order is by increasing user-assigned element numbers.

PROBLEM 10 - FULLY STRESSED DESIGN EXAMPLE

DATE - 77/09/06.

				REF. FRAME OR SURFACE	DIRECTION VECTOR OR ORIENTATION NODE
1	COVER LOADCASE P(PIU) AIRLOAD 3.5000000	1 12 11 P(P2U) P(P3U) P(P4U)	P(P1L) P(P2L) P(P3L) P(P4L)	GLOBAL	0.000000 0.000000 1.000000
2	COVER LOADCASE P(PIU) AIRLOAD 3.0000000	1 2 13 12 P(P2U) P(P3U) P(P4U)	P(P1L) P(P2L) P(P3L) P(P4L)	GLOBAL	0.000000 0.000000 1.000000
3	COVER LOADCASE P(PIU) AIRLOAD 2.5000000	2 3 14 13 P(P2U) P(P3U) P(P4U)	P(P1L) P(P2L) P(P3L) P(P4L)	GLOBAL	0.000000 0.000000 1.000000
4	COVER LOADCASE P(PIU) AIRLOAD 2.0000000	3 4 15 14 P(P2U) P(P3U) P(P4U)	P(P1L) P(P2L) P(P3L) P(P4L)	GLOBAL	0.000000 0.000000 1.000000
5	COVER LOADCASE P(PIU) AIRLOAD 1.5000000	4 5 16 15 P(P2U) P(P3U) P(P4U)	P(P1L) P(P2L) P(P3L) P(P4L)	GLOBAL	0.000000 0.000000 1.000000
6	COVER LOADCASE P(PIU) AIRLOAD 1.0000000	5 6 17 16 P(P2U) P(P3U) P(P4U)	P(P1L) P(P2L) P(P3L) P(P4L)	GLOBAL	0.000000 0.000000 1.000000
7	COVER LOADCASE P(PIU) AIRLOAD .5000000	6 7 17 P(P2U) P(P3U) P(P4U)	P(P1L) P(P2L) P(P3L) P(P4L)	GLOBAL	0.000000 0.000000 1.000000

NOTES:

1. Output from PRINT INPUT (LOADS) statement. Line 14 of Control Program (sec. 8.8.2).
2. The input element loads data are shown by lines 35-40 in section 8.8.3.
3. Printout of the input element-distributed loads is in an increasing element-number order.
4. Only the pressure intensities that are specified for the points corresponding to the element types are printed.
5. Multiple loadcases with distributed loads would all be shown for each element.

*** ELEMENT THERMAL LOADS *** SET NO. 1 STAGE NO. 1

PROBLEM ID -- FULLY STRESSED DESIGN EXAMPLE

1	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	1 P1L	2 -200.0000	3 -200.0000	4 -200.0000	5 -200.0000	6 -200.0000	7 -200.0000	8 -200.0000	9 -200.0000	10 -200.0000	11 -200.0000	12 -200.0000	13 -200.0000	14 -200.0000	15 -200.0000	16 -200.0000	17 -200.0000	18 -200.0000	19 -200.0000	20 -200.0000	21 -200.0000	22 -200.0000	23 -200.0000	24 -200.0000	25 -200.0000	26 -200.0000	27 -200.0000	28 -200.0000	29 -200.0000	30 -200.0000	31 -200.0000	32 -200.0000	33 -200.0000	34 -200.0000	35 -200.0000	36 -200.0000	37 -200.0000	38 -200.0000	39 -200.0000	40 -200.0000	41 -200.0000	42 -200.0000	43 -200.0000	44 -200.0000	45 -200.0000	46 -200.0000	47 -200.0000	48 -200.0000	49 -200.0000	50 -200.0000	51 -200.0000	52 -200.0000	53 -200.0000	54 -200.0000	55 -200.0000	56 -200.0000	57 -200.0000	58 -200.0000	59 -200.0000	60 -200.0000	61 -200.0000	62 -200.0000	63 -200.0000	64 -200.0000	65 -200.0000	66 -200.0000	67 -200.0000	68 -200.0000	69 -200.0000	70 -200.0000	71 -200.0000	72 -200.0000	73 -200.0000	74 -200.0000	75 -200.0000	76 -200.0000	77 -200.0000	78 -200.0000	79 -200.0000	80 -200.0000	81 -200.0000	82 -200.0000	83 -200.0000	84 -200.0000	85 -200.0000	86 -200.0000	87 -200.0000	88 -200.0000	89 -200.0000	90 -200.0000	91 -200.0000	92 -200.0000	93 -200.0000	94 -200.0000	95 -200.0000	96 -200.0000	97 -200.0000	98 -200.0000	99 -200.0000	100 -200.0000
1	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	1 P1L	2 -200.0000	3 -200.0000	4 -200.0000	5 -200.0000	6 -200.0000	7 -200.0000	8 -200.0000	9 -200.0000	10 -200.0000	11 -200.0000	12 -200.0000	13 -200.0000	14 -200.0000	15 -200.0000	16 -200.0000	17 -200.0000	18 -200.0000	19 -200.0000	20 -200.0000	21 -200.0000	22 -200.0000	23 -200.0000	24 -200.0000	25 -200.0000	26 -200.0000	27 -200.0000	28 -200.0000	29 -200.0000	30 -200.0000	31 -200.0000	32 -200.0000	33 -200.0000	34 -200.0000	35 -200.0000	36 -200.0000	37 -200.0000	38 -200.0000	39 -200.0000	40 -200.0000	41 -200.0000	42 -200.0000	43 -200.0000	44 -200.0000	45 -200.0000	46 -200.0000	47 -200.0000	48 -200.0000	49 -200.0000	50 -200.0000	51 -200.0000	52 -200.0000	53 -200.0000	54 -200.0000	55 -200.0000	56 -200.0000	57 -200.0000	58 -200.0000	59 -200.0000	60 -200.0000	61 -200.0000	62 -200.0000	63 -200.0000	64 -200.0000	65 -200.0000	66 -200.0000	67 -200.0000	68 -200.0000	69 -200.0000	70 -200.0000	71 -200.0000	72 -200.0000	73 -200.0000	74 -200.0000	75 -200.0000	76 -200.0000	77 -200.0000	78 -200.0000	79 -200.0000	80 -200.0000	81 -200.0000	82 -200.0000	83 -200.0000	84 -200.0000	85 -200.0000	86 -200.0000	87 -200.0000	88 -200.0000	89 -200.0000	90 -200.0000	91 -200.0000	92 -200.0000	93 -200.0000	94 -200.0000	95 -200.0000	96 -200.0000	97 -200.0000	98 -200.0000	99 -200.0000	100 -200.0000
2	COVER LOADCASE THERMAL	MATL 1 PIU	ELEM TEMP P2U P3U	70 P3U	NODES P4U	1 P1L	2 -200.0000	3 -200.0000	4 -																																																																																																

NOTES:

1. Output from PRINT INPUT (LOADS) statement. Line 14 of Control Program (sec. 8.8.2).
2. The input element thermal loads data are shown by lines 41-45 in section 8.8.3.
3. Similar format as shown on the preceding page for the input element distributed loads. In this printout, however, the nodal delta-temperatures are shown.

INPUT DESIGN ELEMENT DATA FOR SET NO. 1															PAGE NO. 1									
*****															*****									
INTERNAL * USER * TYPE * PROPERTY DATA * UPPER BOUND DATA * LOWER BOUND DATA * FIXED DATA * MARGIN DATA															*****									
NO * NO *																								

NOTES:

- Output from PRINT INPUT (DESIGN) statement. Line 15 of Control Program (sec. 8.8.2).
- The input design data are shown by lines 55-77 in section 8.8.3.
- Design Property data and the specified resizing constraints are printed.

*** BOUNDARY CONDITION DATA ***
 DATA SET NUMBER 1, EXECUTION STAGE 1 DATE 77/09/06 PAGE 1
 PROBLEM 10 -- FULLY STRESSED DESIGN EXAMPLE

COORDINATE SYSTEM REFERENCE TABLE

SYSTEM (USERID)	TYPE	FREEDOM LABELS						FREEDOM-FORCE LABELS					
		1	2	3	4	5	6	1	2	3	4	5	6
1. GLOBAL	REC												
		TX	TY	TZ	RX	RY	AZ	FX	FY	FZ	MX	MY	MZ
2. SPAN	REC												
		TX	TY	TZ	RX	RY	AZ	FX	FY	FZ	MX	MY	MZ

THE CODE IN THE FOLLOWING TABLE IS AS FOLLOWS

CHARACTER F DENOTES A PARTITION 1 FREEDOM (F = FREE)
 CHARACTER R DENOTES A PARTITION 2 FREEDOM (R = RETAIN)
 CHARACTER S DENOTES A PARTITION 3 FREEDOM (S = SUPPORT)
 A BLANK DENOTES A FREEDOM WITH ZERO STIFFNESS

USER NODE	INTERNAL NODE	INPUT ORDER	FREEDOM CODE						ANALYSIS REF. FRAME
			1	2	3	4	5	6	
1	13	2	S	S	S	S	S	S	SPAN
2	11	3	F	F	F	F	F	F	SPAN
3	10	4	F	F	F	F	F	F	SPAN
4	8	5	F	F	F	F	F	F	SPAN
5	6	6	F	F	F	F	F	F	SPAN
6	4	7	F	F	F	F	F	F	SPAN
7	2	8	F	F	F	F	F	F	SPAN
11	16	9	S	S	S	S	S	S	SPAN
12	15	10	F	F	F	F	F	F	SPAN
13	12	11	F	F	F	F	F	F	SPAN
14	9	12	F	F	F	F	F	F	SPAN
15	7	13	F	F	F	F	F	F	SPAN
16	5	14	F	F	F	F	F	F	SPAN
17	3	15	F	F	F	F	F	F	SPAN
20	1	16	S	S	S	S	S	S	SPAN
101	14	17	S	S	S	S	S	S	SPAN
111	17	17	S	S	S	S	S	S	SPAN

THE STIFFNESS MODULE HAS BEEN EXECUTED FOR THIS DATA SET AND STAGE.

NOTES:

1. Output from PRINT INPUT (BC) statement. Line 17 of Control Program (sec. 8.8.2).
2. The input BC data are shown by lines 24-26 in section 8.8.3.
3. Default printout order is by increasing user-assigned node numbers.
4. Inactive freedoms are identified by blanks in the "freedom-code" columns. For this to be shown in the BC printout, execution of the STIFFNESS Processor must have been done previously (e.g. by the PERFORM DESIGN statement).

USER ID

1	COVER	INPUT RECORD	17,	INTERNAL NO.	21,	TEMPERATURE	70,	MATERIAL CODE	1
		MATERIAL ALLOWABLE	M 2 ;	COMPRESSION TABLE	NONE ;	SHEAR TABLE	NONE ;	2-DIRECTION FACTOR	1.000
		USER NODES	INTERNAL NODES			PROPERTIES			
		1	13			T(0)U =	.09409		
		12	15			T(1)U =	0.00000		
		11	16			T(2)U =	0.00000		
						ALPHAU =	0.00000		
						BETAU =	0.30000		
						T(0)I =	.09291		
						T(1)I =	0.00000		
						T(2)I =	0.00000		
						ALPHAI =	0.00000		
						BETAI =	0.00000		

2	COVER	INPUT RECORD	18,	INTERNAL NO.	17,	TEMPERATURE	70,	MATERIAL CODE	1
		MATERIAL ALLOWABLE	M 2 ;	COMPRESSION TABLE	NONE ;	SHEAR TABLE	NONE ;	2-DIRECTION FACTOR	1.000
		USER NODES	INTERNAL NODES			PROPERTIES			
		1	13			T(0)U =	.06961		
		2	11			T(1)U =	0.00000		
		13	12			T(2)U =	0.00000		
		12	15			ALPHAU =	0.00000		
						BETAU =	0.00000		
						T(0)I =	.08541		
						T(1)I =	0.00000		
						T(2)I =	0.00000		
						ALPHAI =	0.00000		
						BETAI =	0.00000		

3	COVER	INPUT RECORD	16,	INTERNAL NO.	14,	TEMPERATURE	70,	MATERIAL CODE	1
		MATERIAL ALLOWABLE	M 2 ;	COMPRESSION TABLE	NONE ;	SHEAR TABLE	NONE ;	2-DIRECTION FACTOR	1.000
		USER NODES	INTERNAL NODES			PROPERTIES			
		2	11			T(0)U =	.08669		
		3	10			T(1)U =	0.00000		
		14	9			T(2)U =	0.00000		
		13	12			ALPHAU =	0.00000		
						BETAU =	0.00000		
						T(0)I =	.11018		
						T(1)I =	0.00000		
						T(2)I =	0.00000		
						ALPHAI =	0.00000		
						BETAI =	0.00000		

NOTES:

1. Output from PRINT OUTPUT (DESIGN) statement. Line 19 of Control Program (sec. 8.8.2).
2. The new element properties, as resized by the DESIGN Processor, are printed. The gages shown here are those resulting from 5 design cycles. These data can be compared to the initial gages.

***** OUTPUT DESIGN HISTORY DATA - MINIMUM MARGINS OF SAFETY FOR SET NO. 1 *****										***** PAGE NO. 1 *****									
***** INTERNAL * USER * ELEMENT * MS *****										***** MARGINS OF SAFETY FOR USER SPECIFIED CYCLES - 10 CYCLES PER LINE *****									
***** ELEMENT * ELEMENT * TYPE * LABEL *****										*****									
1	*	15	*	SPAR	*	MSTWB	61.859	.987	11.672	12.233	7.144								
1	*	15	*	SPAR	*	MSA-CAP1U	88.835	-.552	1.154	.025	.353								
1	*	15	*	SPAR	*	MSA-CAP1L	61.655	-.642	.792	-.074	.122								
1	*	15	*	SPAR	*	MSA-CAP2U	88.835	-.552	1.154	.025	.353								
1	*	15	*	SPAR	*	MSA-CAP2L	61.655	-.642	.792	-.074	.122								
1	*	15	*	SPAR	*	MSASTI	69.487	-.956	-.517	-.866	-.822								
2	*	30	*	SPAR	*	MSTWB	52.879	1.044	13.674	14.439	7.376								
2	*	30	*	SPAR	*	MSA-CAP1U	82.487	-.678	.925	-.021	.241								
2	*	30	*	SPAR	*	MSA-CAP1L	118.286	-.749	.655	-.086	.081								
2	*	30	*	SPAR	*	MSA-CAP2U	82.487	-.678	.925	-.021	.241								
2	*	30	*	SPAR	*	MSA-CAP2L	118.286	-.749	.655	-.086	.081								
2	*	30	*	SPAR	*	MSASTI	92.938	-.961	-.294	-.839	-.751								
3	*	7	*	COVER	*	MST	1000.000	-.997	1.065	-.314	.010								
3	*	7	*	COVER	*	MST1	1000.000	1000.000	1000.000	1000.000	1000.000								
3	*	7	*	COVER	*	MST2	1000.000	1000.000	1000.000	1000.000	1000.000								
3	*	7	*	COVER	*	MST	1000.000	-.996	1.200	-.244	-.100								
3	*	7	*	COVER	*	MST1	1000.000	1000.000	1000.000	1000.000	1000.000								
3	*	7	*	COVER	*	MST2	1000.000	1000.000	1000.000	1000.000	1000.000								
4	*	6	*	COVER	*	MST	1000.000	-.998	1.063	-.279	.172								
4	*																		

NOTES:

1. Output from PRINT OUTPUT (DESIGN) statement. Line 20 of Control Program (sec. 8.8.2).
2. The "history" printout shows how the minimum element-property margins of safety vary as multiple design cycles are performed (in this case, 5 design cycles were executed).

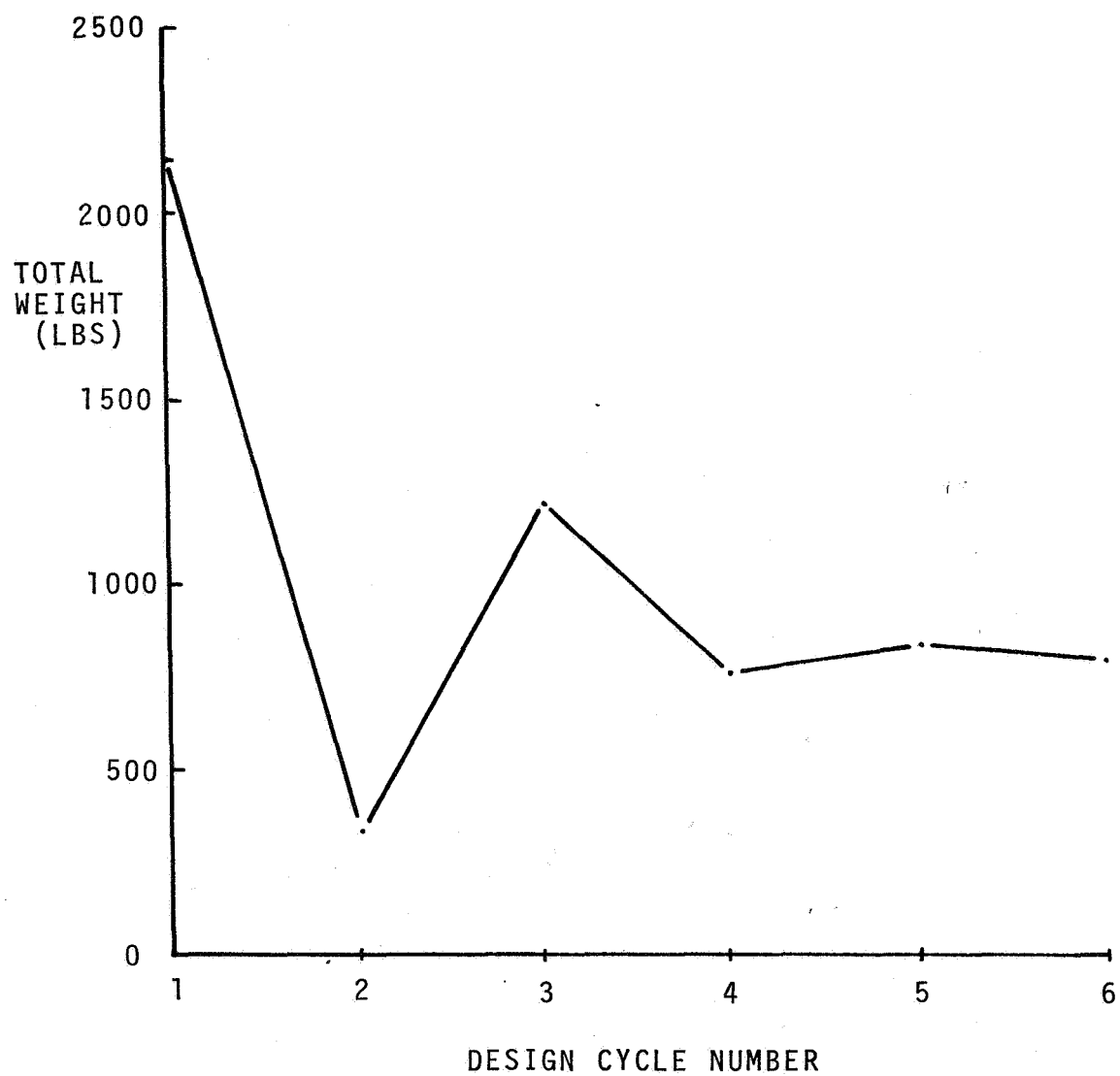


Figure 8-8. Total Weight vs. Design Cycle Number

9.0 SNARK LANGUAGE

To simplify the solution of matrix related problems not directly solvable by the integrated capabilities of the ATLAS System, the high level matrix-oriented language, SNARK, is provided by ATLAS. The SNARK package includes the SNARK language, the SNARK precompiler, and the SNARK library support routines. It is used extensively throughout the ATLAS System and can be used within ATLAS control decks to perform many matrix operations including algebraic matrix manipulation, matrix input and output on either random or sequential files, management of matrix storage in core, and management of the ATLAS data base. Table 9-1 shows a summary of the SNARK commands. Detailed descriptions of SNARK are presented in the SNARK User's Manual ATLAS documentation.

9.1 CONTROL PROGRAM APPLICATIONS

The ATLAS Control Program, as created from the user-written execution-control data deck, provides capabilities for performing the following functions:

- a) Define the sequence and mode of execution of selected ATLAS modules
- b) Control ATLAS data-management functions
- c) Perform special analyses not directly supported by the ATLAS System
- d) Perform data interfaces between the ATLAS System and external programs

The first two functions are supported by the ATLAS System execution-control language whereas, in general, the last two functions require the use of SNARK and FORTRAN. The primary function of SNARK in performing special analysis and interface capabilities is in the area of data management and matrix algebra. These functions are discussed in the following sections.

9.1.1 Data Management

Whenever user defined capabilities are defined in an ATLAS control deck, the need to manipulate data within the ATLAS data base is present. This may involve simply reading a data matrix, adding data to the data base, or modifying the existing data and replacing it in the data base. The reading and replacement of data matrices, and the in-core management of the accessed data

are handled by SNARK. The SNARK command RDMATRB, for example, is used to access a specific data matrix within the ATLAS data base. The matrix is read into core and stored in blank common with any other data matrices read or defined by the Control Program. All data matrices are stored in blank common between the last address loaded and the start of the random access tables. This storage area is managed by SNARK in such a way as to prevent wasted space and to eliminate the need for the user to know where, in blank common, a given matrix is located. The user-assigned position number of a data matrix does not necessarily correspond to its position in core. When matrices change size, or are written out, created, or deleted, SNARK may reshuffle the position of matrices in core. This reshuffling is invisible to the user, as all references to a matrix are made with the position reference number which SNARK automatically relates to the actual matrix position in blank common.

Adding or replacing data in the data base is accomplished by using the SNARK command WTMATRB. This command allows a data matrix to be moved from blank common to the specified portion of the ATLAS data base.

In addition to managing the ATLAS data base input and output, SNARK can be used to access and/or replace data items within a data matrix in core. This accessing can be done with a number of SNARK commands including SETELEM, SNARK CALL, and MOVE. The SETELEM command is used to set a matrix element or elements to a specified value, or to set a FORTRAN variable to the value of a matrix element. This command is very inefficient for large blocks of data. In this case, it is more efficient to pass the data matrix to a FORTRAN subroutine using the SNARK CALL command. The subroutine can then use the data matrix just as if it had received a FORTRAN array. Another method for using data matrices as FORTRAN arrays is to move the data from blank common to a dimensioned FORTRAN array using the SNARK MOVE command. These SNARK commands allow the user to access individual data items or portions of data. SNARK also supports a group of commands which allow the user to manipulate directly the data matrices in blank common. These commands include the matrix arithmetic commands discussed in the following section, as well as commands for deleting rows or columns, extracting rows or columns, extracting partitions, and merging matrices either rowwise or columnwise.

9.1.2 Matrix Arithmetic

Direct manipulation of data matrices stored in blank common is supported by the SNARK language with a set of matrix arithmetic commands. These commands (table 9-1) allow the user to add, subtract, multiply and invert ATLAS data matrices that contain real numbers and are stored as a single partition.

Table 9-1. Summary of the SNARK Commands

COMMAND	DESCRIPTION
DATA MANIPULATION COMMANDS	
DEFID	Define a matrix at a specified position
EXTID	Obtain a matrix row and column dimension and type
CHGMID	Change the dimension or type of a matrix
CLEARMAT	Set all elements of matrix to zero
CLEARPOS	Clear a matrix position and release its storage area
STOAUX	Store specified data in the matrix auxiliary ID
EXTAUX	Extract the data in the matrix auxiliary ID
DELCLS DELROWS	Creates a new matrix from an existing one by deleting specified rows or columns
EXTCLS EXTRWS	Creates new matrix from an existing one by extracting specified rows or columns
EXTPRT	Extracts a specified partition and stores it in a second matrix
MRGCLS MRGRWS	Used to merge two matrices rowwise or columnwise
MOVE	Moves elements between SNARK matrices and FORTRAN arrays
SETELEM	Sets matrix elements to a value or sets a FORTRAN variable to a matrix element
SNARK CALL	Call a FORTRAN subroutine or function

Table 9-1. (Concluded)

COMMAND	DESCRIPTION
INPUT/OUTPUT COMMANDS	
PRNTMAT	Prints a matrix
PRNTMID	Prints a matrix ID (header record)
RDMATRB	Reads a matrix from a random binary file
RDMATSB	Reads a matrix from a sequential binary file
WTMATRB	Writes a matrix on a random binary file
WTMATSB	Writes a matrix on a sequential binary file
MATRIX ARITHMETIC COMMANDS	
ADD	Adds two matrices together
ADDT	Adds a matrix to the transpose of another
SUB	Subtracts one matrix from another
SUBT	Subtracts the transpose of one matrix from another
MPY	Multiplies two matrices together
MPYT	Postmultiplies a matrix by the transpose of another
TMPY	Premultiplies a matrix by the transpose of another
INVERT	Inverts a square matrix
TPSE	Transposes a matrix
PROGRAMMING AID COMMANDS	
INVENTORY	Prints an inventory of the currently defined matrix positions
MATRIX	Declares a SNARK program or subroutine
SETERR	Controls the processing of SNARK errors
STOINV	Obtain information about the data storage area (Blank Common)

9.2 EXTERNAL PROGRAM INTERFACES

The capability to perform data interfaces between the ATLAS System and external programs is supported by SNARK, in conjunction with FORTRAN and the ATLAS execution-control language. When an interface between ATLAS and an external program is required, the general procedure is to use ATLAS to generate the required data and to store the data in the ATLAS data base. SNARK language commands, in conjunction with FORTRAN, are then used to access the data from the ATLAS data base, reformat it as necessary, and write the reformatted data onto the file or files required by the external program. Data generated by an external program can be interfaced with ATLAS by using SNARK commands and FORTRAN statements within an ATLAS Control Program to read the data, format it into the required ATLAS internal format, and write the data into the ATLAS data base where it can be used for any subsequent processing by the ATLAS System. In general, the data interface capabilities of the system are unlimited and may involve card input data, ATLAS generated data, externally generated data on random or sequential files, or data generated by an ATLAS Control Program.

Figure 9-1 shows a flow diagram of an interface between ATLAS and the FLEXSTAB system (ref. 9-1) providing ATLAS generated mass and flexibility data to FLEXSTAB, and steady state loads data from FLEXSTAB to ATLAS for performing stress analysis and structural design functions. This interface is one of a series of standard ATLAS interfaces available to the user. Other standard interfaces, as described in appendix G of the ATLAS User's Manual, include:

- a) NASA-LRC AIRPLANE CONFIGURATION PROGRAM INTERFACE--Interface of the geometry input data for the NASA-LRC aerodynamic configuration programs (ref. 9-2) with ATLAS. ATLAS nodal data can be defined by use of the interfaced geometry data.
- b) ATLAS/NASTRAN SYSTEM INTERFACES--Interface of ATLAS input data to the NASTRAN System (ref. 9-3) and an interface of NASTRAN input data to ATLAS.

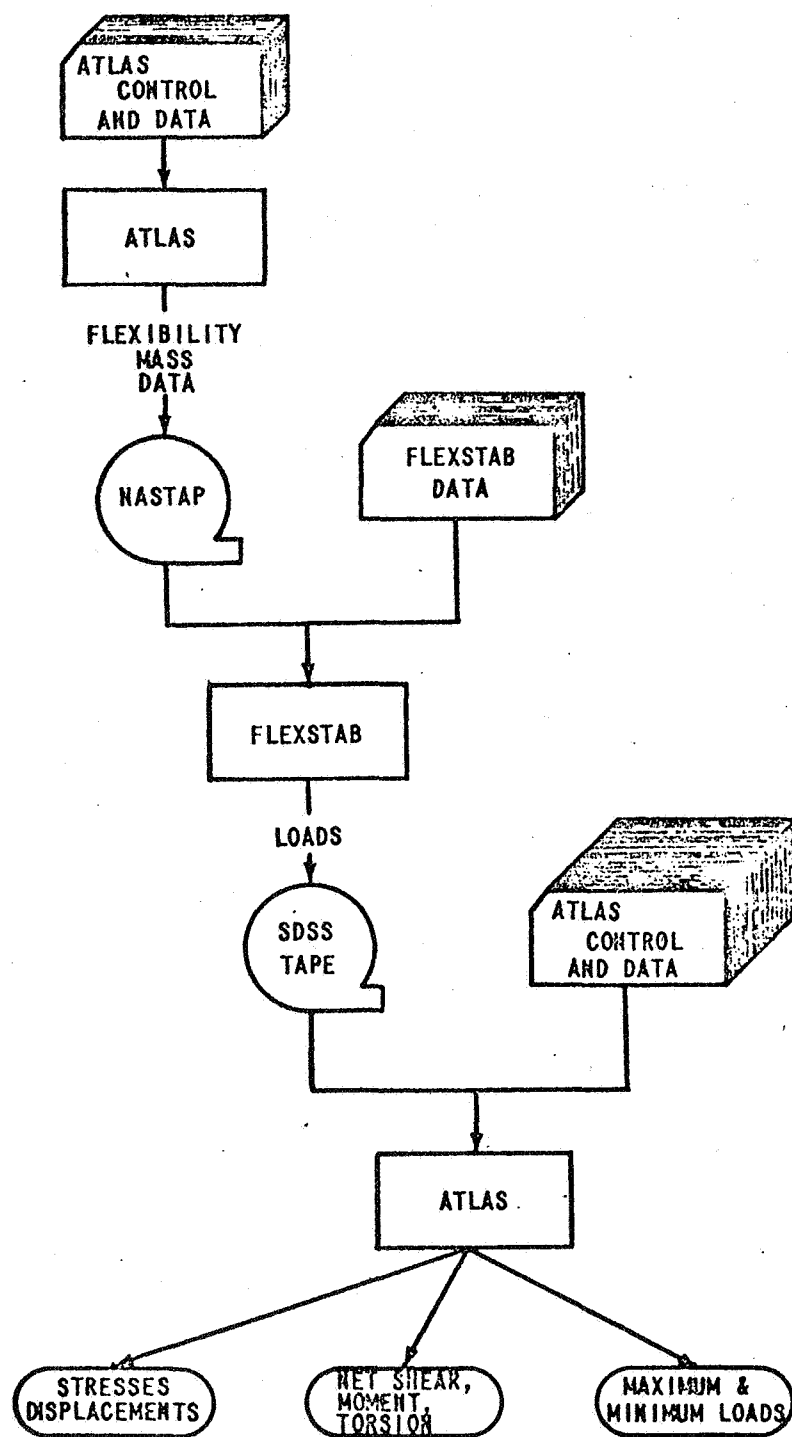


Figure 9-1. Schematic of ATLAS/FLEXSTAB System Interfaces

9.2.1 Control Program Example

```

C      MATRIX SUBROUTINE STREFIL
C
C      PURPOSE  THIS ROUTINE READS THE FLEXSTAB SYMMETRIC NET LOADS
C                FROM FILE SDSS AND SETS UP THE ATLAS NODAL LOADS MATRICES
C
C      INPUT    THE SDSS TAPE FROM FLEXSTAB
C
C                (REF) D6-41064-3 FLEXSTAB 1.02.00 PROGRAM DESCRIPTION)
C
C      OUTPUT   QSYMMXX - SYMMETRIC NODAL LOADS MATRIX
C
C                NOTE  THE LOAD CASE NUMBERS WILL BE ASSIGNED
C                      SEQUENTIALLY BEGINING WITH THE FIRST
C                      INPUT FILE.
C
C      FILE USAGE - DATARNF
C                   SAVESSF
C
C      COMMON USAGE -
C
C      XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
C      X          SYSTEM  ERRORS  AND  WARNINGS  X
C      X
C      X          / K E R R O R /
C      X
C      XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
C
C      COMMON /KERROR/
C      *      KERROR,KWARN,IPTU
C      DIMENSION KERRORS(1)
C      EQUIVALENCE (KERRORS,KWARN)
C
C      XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
C      X          PROGRAM  SEQUENTIAL  FILES  X
C      X
C      X          / K Q B U F P /
C      X
C      XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
C
C      COMMON /KQBUFP/  KQBUFP(2),
C      *      SAVESSF , SAVESS1 , SAVESS2 , SAVESS3 , SAVESS4 ,
C      *      SC00SSF , SC01SSF , SC02SSF , SC03SSF , SC04SSF ,
C      *      DBASFIL
C      INTEGER SAVESSF , SAVESS1 , SAVESS2 , SAVESS3 , SAVESS4 ,
C      *      SC00SSF , SC01SSF , SC02SSF , SC03SSF , SC04SSF ,
C      *      DBASFIL
C
C      XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
C      X          PROGRAM  RANDUM  FILES  X
C      X
C      X          / K Q R N D M /
C      X
C      XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
C
C      COMMON /KQRNDM/

```

```

*      KQINDX , KQRNDN , KQRNDI , KQRND ,
*      ADDIRNF , AF10RNF , CHOLRNF , DATARNF , DESIRNF ,
*      DUBLRNF , INTERNF , LOADRNF , MACHRNF , MASSRNF ,
*      MERGRNF , MULTRNF , RHO3RNF , STIRNF , STRERNF ,
*      CONTRNF , VIBRRNF , FLUTRNF , FLEXRNF , EXTRRNF ,
*      FLORRNF , SEARRNF , SCALRNF , BUCKRNF , ALGERNF ,
*      FUELRNF , PAYLRNF , RESERVE(3) ,
*      SCOORNF , SC01RNF , SC02RNF , SC03RNF , SC04RNF ,
*      SC00RIF , SC01RIF , SC02RIF , SC03RIF , SC04RIF
C
      INTEGER ADDIPNF , AF10RNF , CHOLRNF , DATARNF , DESIRNF ,
*      DUBLRNF , INTERNF , LOADRNF , MACHRNF , MASSRNF ,
*      MERGRNF , MULTRNF , RHO3RNF , STIRNF , STRERNF ,
*      CONTRNF , VIBRRNF , FLUTRNF , FLEXRNF , EXTRRNF ,
*      FLORRNF , SEARRNF , SCALRNF , BUCKRNF , ALGERNF ,
*      FUELRNF , PAYLRNF , RESERVE(3) ,
*      SCOORNF , SC01RNF , SC02RNF , SC03RNF , SC04RNF ,
*      SC00RIF , SC01RIF , SC02RIF , SC03RIF , SC04RIF
C
      LOGICAL IFRREC
C
C      SET UP FILE CATALOG ARRAY - 2*50 (50 LOAD CASES)
C
      DIMENSION CATLOG(2,50),IDUM(10)
      INTEGER CATLOG
C
C      INIALIZE POINTERS AND SET UP THE MATRIX CATALOG
C
      DO 99 K=1,100
99  CATLOG(K) = 0
      NC = 1
      REWIND SAVESSF
100 READ (SAVSSF) NAME,IDUM(1),IDUM(2),NFILE
      IF(EOF(SAVSSF)) 199,105
105 IF(NAME.NE.10H*FINISHED*) GO TO 110
      NC = NC + 1
      GO TO 100
C
110 IF(NAME.EQ.6H(L1)-S) CATLOG(1,NC) = NFILE
      GO TO 100
C
C      REWIND THE SDSS FILE AND SET THE NIMBER OF CASES
C
199 REWIND SAVESSF
      IFILE = 0
      NCASES = NC - 1
C
C      ESTABLISH INDEX NAMES
C
      NQSYMM = 7LQSYMMXX
C
C      DEFINE POSITION REFERENCE
C
      NPOS = 1
C
C      LOOP TO 1000 FOR EACH LOAD CASE
C
      LS = 0

```

```

      DO 1000 K=1,NCASES
C
C      POSITION THE LOADS FILE FOR THE SYMMETRIC MATRICES
C
      NFILE = CATLOG(1,K)
      IF(NFILE.EQ.0) GO TO 300
      IF(NFILE-IFILE) 210,20,220
210  REWIND SAVESSF
      IFILE = 0
220  ISKIP = NFILE - IFILE
      DO 230 L=1,ISKIP
225  READ (SAVESSF)
      IF(EOF(SAVESSF)) 230,225
230  CONTINUE
      IFILE = NFILE
C
C      SYMMETRIC MATRIX
C
20  READ (SAVESSF) (IDUM(J),J=1,10)
      NROW = IDUM(8)
      IF(NROW.EQ.0) GO TO 1000
      IF(LS.GT.0) GO TO 21
      NS = NROW + 1
      DEFID NPOS,NS,NCASES,MIXED
      NROS = NROW
21  LS = LS + 1
      IF(NROW.NE.NROS) GO TO 2000
      SETELEM NPOS(1,LS) TO =LS
C
C      CALL LREAD TO READ THE LOADS FILE
C
      SNARK CALL LREAD(NPOS,=LS,=NA,=SAVESSF)
C
C      SPACE TO THE NEXT FILE
C
999  READ (SAVESSF)
      IF(EOF(SAVESSF)) 1000,999
1000 CONTINUE
C
C      OUTPUT THE SYMMETRIC MATRICES
C
      IF(LS.EQ.0) GO TO 1600
      CHGMID NPOS,NS,LS,MIXED,SYMMETRIC
      PRNTMAT -1L,NPOS,SYMMETRIC NODAL LOADS MATRIX FROM FLEXSTAB
      WTMATR8 DATARNE,NQSYMM,NPOS
      GO TO 2500
C
C      ERROR MESSAGE
C
2000 PRINT 2001
2001 FORMAT(///10X,* ERROR - INCOMPATIBLE ROW DIMENSIONS FOR THE FLEXST
1AB NET LOADS - PROCESSING ABORTED *///)
      KERROR = KERROR + 1
C
2500 CONTINUE
      RETURN
      END

```

```

C      MATRIX SUBROUTINE LREAD(APOS,LS,NA,SAVESSF)
C
C      THIS SUBROUTINE READS THE FLEXSTAB LOADS FILE AND STORES
C      THE LOADS IN BLANK COMMON (MATRIX QSYMMXX)
C
      DIMENSION APOS(NA,1)
      READ (SAVESSF) (APOS(J,LS),J=2,NA)
      RETURN
      END

```

9.3 SPECIAL ANALYSIS CAPABILITY

SNARK may be used in conjunction with FORTRAN to define analysis and/or input/output capabilities that are not directly provided by the ATLAS System. These "special" Control Programs can be used simply to reformat output, or they may be complete analysis programs using ATLAS as an input generator and data management system. For these applications, SNARK is generally used to handle the required data management and matrix arithmetic functions.

Figure 9-2 illustrates a simplified procedural outline for defining additional analysis capabilities within an ATLAS Control Program. This procedure includes capabilities defined strictly within the Control Program, and the option to perform part of the analysis external to the ATLAS System. In this respect, the procedural outline may apply to both external program interfaces and special analysis capabilities within a Control Program.

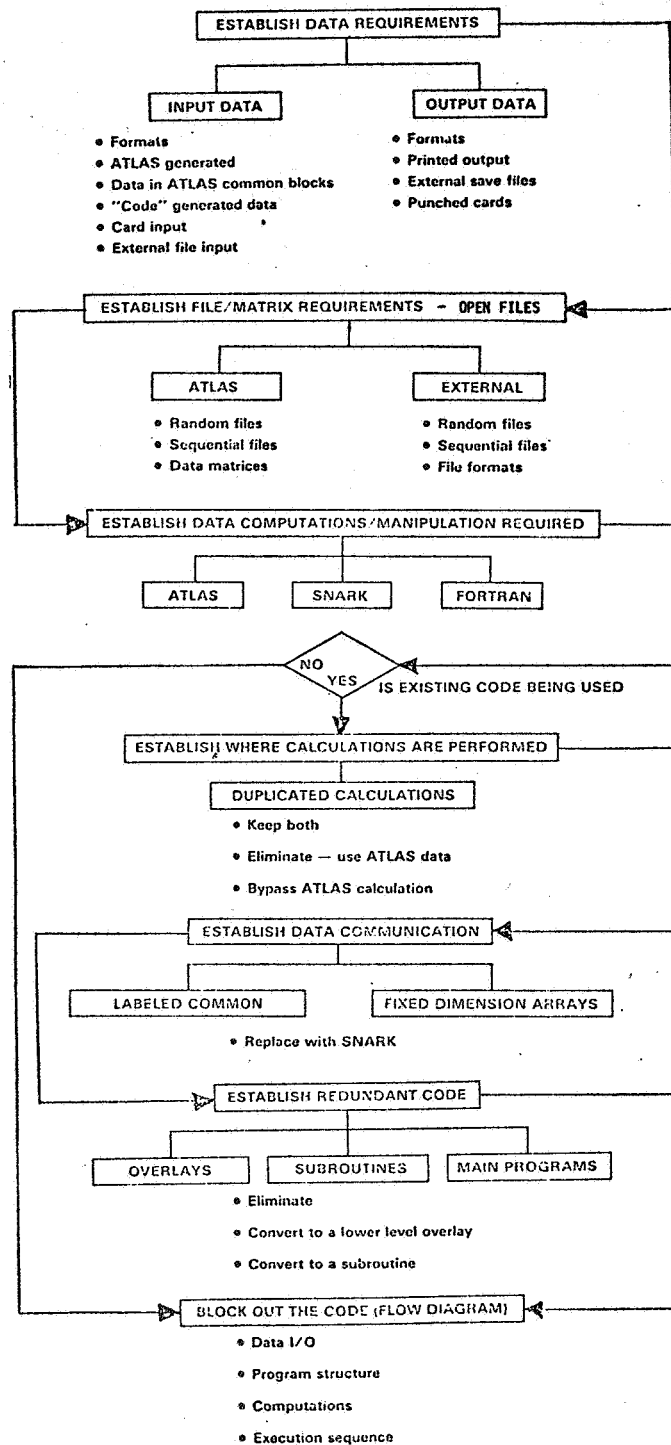


Figure 9-2. Procedural Outline for Control Program Coding

9.3.1 Control Program Example

```
C
C BEGIN CONTROL MATRIX PROGRAM DAVE
C
C THIS CONTROL PROGRAM PRESENTS AN EXAMPLE OF
C A SPECIAL ANALYSIS CAPABILITY THAT ALLOWS THE
C USER TO SPECIFY THAT CERTAIN ROD ELEMENTS CAN ONLY
C CARRY COMPRESSION (OR TENSION) LOADS. IF A
C COMPRESSION (TENSION) ONLY ELEMENT IS IN
C TENSION (COMPRESSION) THE CROSS SECTIONAL AREAS
C WILL BE SET TO A USER SPECIFIED SMALL VALUE,
C EFFECTIVELY REMOVING THE ELEMENT FROM THE
C STRUCTURE. THE PROGRAM WILL AUTOMATICALLY CYCLE
C UNTIL NO ELEMENT CHANGES TAKE PLACE.
C
C COMMON /USERCOM/LCS,ICYC,KCYC,NSET,NSTAGE
C COMMON /CARDS/REC(250),ID(250),KRECS,KCDS,ITEMS,KERR(3),JBL
C DIMENSION FET(100,3),IREC(250)
C DATA NOUT/6/
C EQUIVALENCE (REC,IREC)
C PROBLEM ID(TENSION/COMPRESSION ONLY PROGRAM - ROD ELEMENTS)
C
C READ ATLAS DATA
C
C READ INPUT
C
C READ TENSION COMPRESSION DATA - STACKED AFTER END PROBLEM DATA
C
C FORMAT -- BEGIN COMPRESSION DATA /
C           CYCLES N LC M /
C           NSUB CVAL TVAL /
C           END COMPRESSION DATA /
C
C           WHERE - N = NUMBER OF CYCLES
C                   M = LOAD CASE IDENTIFICATION
C                   NSUB = ELEMENT SUBSET NUMBER (INTEGER) OF
C                           SUBSET IDENTIFYING RODS TO BE CHECKED
C                   CVAL = VALUE TO WHICH ROD A1,A2 WILL BE SET
C                           WHEN STRESS IS NEGATIVE (COMPRESSION)
C                   TVAL = VALUE TO WHICH ROD A1,A2 WILL BE SET
C                           WHEN STRESS IS POSITIVE (TENSION)
C
C OPEN THE REQUIRED ATLAS FILES
C
C CALL FILEADD(FET,DATARNF,STRERNF,CONTRNF)
C
C INITIALIZE
C
C NSET=1
C NSTAGE=1
```

```

      ICYC=1
      KCYC=0
      LCS=0

C
C      READ THE STIFFNESS PARAMETER MATRIX,EXTRACT
C      THE NUMBER OF ELEMENTS AND DEFINE THE
C      TENSION/COMPRESSION PROPERTY MATRIX.
C

      RDMATRB DATARNF,7LKPARMS1,1
      SETELEM =NCOL TO 1(13,1)
      DEFID 2,3,NCOL,MIXED
      CLEARPOS 1

C
C      READ AND CHECK THE INPUT DATA
C

      CALL LODAREC
      KERROR = KERROR + KERR(1) + KERR(2)
      IF((IREC(2).AND.77777777000000000000B).NE.4LCOMP) GO TO 8001
10  CALL LODAREC
      KERROR = KERROR + KERR(1) + KERR(2)
      IF(IREC(1).EQ.3HEND) GO TO 90
      IF(IREC(1).NE.6HCYCLES) GO TO 20
      IF(ID(2).NE.5HFIXED) GO TO 8001
      IF(IREC(2).LT.0.OR.IREC(2).GT.100) GO TO 8001
      ICYC = IREC(2)
      IF(IREC(3).NE.2HLC) GO TO 8001
      LCS = IREC(4)
      GO TO 10
20  IF(ID(1).NE.5HFIXED) GO TO 8001
      IF(ID(2).NE.8HFLOATING) GO TO 8001
      IF(ID(3).NE.8HFLOATING) GO TO 8001
      IF(IREC(1).LT.1.OR.IREC(1).GT.999) GO TO 8001
      CV=REC(2)
      TV=REC(3)

C
C      READ THE REQUIRED SUBSET MATRIX AND CALL CTLOAD
C      TO SET THE ELEMENT PROPERTIES.
C

      ISUB = INCR(7LSEK000A,4,6,IREC(1))
      IZZZZ = ISUB
      IF(IFRREC(DATARNF,ISUB)) 30,8000
30  RDMATRB DATARNF,ISUB,1
      EXTID 1,NRS,NCS,NTYPE
      SNARK CALL CTLOAD(1,2,=NRS,=CV,=TV)
      GO TO 10

C
90  CONTINUE

C
C      COMPRESS THE ZEROS AND WRITE THE TENSION/COMPRESSION
C      MATRIX ON CONTRNF.
C

      SNARK CALL COMPRE(2,=NCOL)
      CHGMID 2,3,NCOL,MIXED,=0
      WTMATRB CONTRNF,7LKCVALUE,2
      CLEARPOS

C
C      PERFORM A STANDARD ATLAS STRESS ANALYSIS
C

```

```

100 CONTINUE
    PURGE FILES(STIRNF,MERGRNF,CHOLRNF,MULTRNF,STRERNF)
    PERFORM STRESS
    PRINT OUTPUT(STRESS)
    KCYC = KCYC + 1
    IF(KCYC.GT.ICYC) GO TO 4000
C
C    REDOPEN THE REQUIRED ATLAS FILES AND ESTABLISH
C    THE LOAD CASE REQUIRED.
C
    CALL FILEADD(FET,DATARNF,STRERNF,CONTRNF)
    RDMATR8 STRERNF,7LDCOORAA,1
    EXTID 1,NPC,NCC,NTYPE
    DO 13 K=1,NCC
        SETELEM =LCX TO 1(1,K)
        IF(LCX.EQ.LCS) GO TO 14
    13 CONTINUE
        GO TO 8001
    14 LCX=K
        CLEARPOS 1
C
C    BRING IN MATRICES FOR STRESS EXTRACTION
C
    KCVALP = 1
    KSTRSP = 2
    KSTCOP = 3
    KSFMTN = 4
    KCVALN = 7LKVALUE
    KSTRSN = 5LST001.OR.SHIFT(NSTAGE,24).OR.SHIFT(NSET,18)
    KSTCON = 5LSCN01.OR.SHIFT(NSTAGE,24).OR.SHIFT(NSET,18)
    KSFMTN = 6LKSFO01.OR.SHIFT(NSET,18)
    IZZZZ = KCVALN
    IF(IFRREC(CONTRNF,KCVALN)) 101,8000
    101 RDMATR8 CONTRNF,KCVALN,KCVALP
        IZZZZ = KSTRSN
        IF(IFRREC(STRERNF,KSTRSN)) 102,8000
    102 RDMATR8 STRERNF,KSTRSN,KSTRSP
        IZZZZ = KSTCON
        IF(IFRREC(STRERNF,KSTCON)) 103,8000
    103 RDMATR8 STRERNF,KSTCON,KSTCOP
        IZZZZ = KSFMTN
        IF(IFRREC(DATARNF,KSFMTN)) 104,8000
    104 RDMATR8 DATARNF,KSFMTN,KSFMTN
        EXTID KCVALP,N1,NOELM,N2
        NSBLK = 1
        NTEST = 0
C
C    CALL CHGPROP TO RESET THE ROD PROPERTIES
C
    150 SNARK CALL CHGPROP(KCVALP,KSTRSP,KSTCOP,KSFMTN,=NOELM,
        1      =NTEST,=NSBLK,=LCX)
C
C    NTEST = 1 NEW STRESS MATRIX REQUIRED
C    NTEST = 2 NEW KSF MATRIX REQUIRED
C    NTEST = 3 CYCLE BACK AND RECALCULATE STRESSES
C    NTEST = 4 DO NOT CYCLE AGAIN
C
    GO TO (1000,2000,3000,4000) NTEST

```

```

1000 KSTRSN = INCR(KSTRSN,3,5,1)
      IZZZZ = KSTRSN
      IF(IFRREC(STRENF,KSTRSN)) 1001,8000
1001 RDMATR8 STRENF,KSTRSN,KSTRSP
      NSBLK = NSBLK + 1
      GO TO 150
C
2000 WTMATR8 DATARNF,KSFMTN,KSFMT8
      KSFMTN = INCR(KSFMTN,4,6,1)
      IZZZZ = KSFMTN
      IF(IFRREC(DATARNF,KSFMTN)) 2001,8000
2001 RDMATR8 DATARNF,KSFMTN,KSFMT8
      GO TO 150
C
3000 WTMATR8 DATARNF,KSFMTN,KSFMT8
      GO TO 100
C
4000 CONTINUE
C
C      PRINT THE FINAL DISPLACEMENTS AND REACTIONS
C
      PRINT OUTPUT(DISPLAC)
      PRINT OUTPUT(REACTIONS)
      GO TO 9999
C
C      ERROR MESSAGES
C
8000 WRITE (NOUT,8010) IZZZZ
      KERROR = KERROR + 1
      GO TO 9999
8001 WRITE (NOUT,8011)
      KERROR = KERROR + 1
8010 FORMAT(/5X,* ERROR - UNDEFINED MATRIX *,A10,/)
8011 FORMAT(/5X,* ERROR - ILLEGAL INPUT DATA */)
C
9999 CONTINUE
      END

      MATRIX SUBROUTINE CTLOAD(NSUB,KCVA,NRS,KCV,KTV)
C
C      THIS SUBROUTINE EXTRACTS THE INTERNAL ELEMENT
C      NUMBERS FROM THE ELEMENT SUBSET MATRIX AND PUTS
C      THE CORRESPONDING TENSION-COMPRESSION AREAS IN
C      THE TENSION-COMPRESSION PROPERTY MATRIX.
C
C      NSUB = ELEMENT SUBSET MATRIX
C      KCVA = ROW DIMENSION OF NSUB
C      KCV  = COMPRESSION AREA
C      KTA  = TENSION AREA
C
      DIMENSION NSUB(1),KCVA(3,1)
      DO 50 K=1,NRS
        IW = NSUB(K)
        DO 30 L=1,60
          IW = SHIFT(IW,1)
          IT = IW.AND.18

```

```

      IF(IT.EQ.0) GO TO 30
      NUM = 60*(K-1)+L
      KCVA(1,NUM) = NUM
      KCVA(2,NUM) = KCV
      KCVA(3,NUM) = KTV
30  CONTINUE
50  CONTINUE
      RETURN
      END

```

MATRIX SUBROUTINE COMPRE(KCVA,NCOL)

```

C
C THIS SUBROUTINE COMPRESSES THE ZEROS OUT OF
C THE FINAL TENSION-COMPRESSION PROPERTY MATRIX.
C
C      KCVA = PROPERTY MATRIX TO BE COMPRESSED
C      NCOL = COLUMN DIMENSION OF KCVA
C
      DIMENSION KCVA(3,1)
      NAC = 0
      DO 50 K=1,NCOL
      IF(KCVA(1,K).EQ.0) GO TO 50
      NAC = NAC + 1
      KCVA(1,NAC) = KCVA(1,K)
      KCVA(2,NAC) = KCVA(2,K)
      KCVA(3,NAC) = KCVA(3,K)
50  CONTINUE
      NCOL = NAC
      RETURN
      END

```

MATRIX SUBROUTINE CHGPROP(CVAL,STRS,INDS,ELEM,NOELM, 1 NTEST,NSBLK,LCS)

```

C
C THIS SUBROUTINE CHECKS THE ROD STRESSES AND IF THE
C ELEMENT IS IN TENSION THE AREA IS SET TO THE
C TENSION AREA AND IF THE ELEMENT IS IN COMPRESSION
C THE AREA IS SET TO THE COMPRESSION AREA.
C
C      CVAL = TENSION-COMPRESSION PROPERTY MATRIX
C      STRS = STRESS MATRIX PARTITION
C      INDS = STRESS CONTROL MATRIX
C      ELEM = STRUCTURAL ELEMENT MATRIX PARTITION
C      NOELM = NUMBER OF ELEMENTS
C      NTEST = PARTITION INDICATOR
C      NSBLK = NUMBER OF STRESS MATRIX PARTITIONS
C      LCS = LOAD CASE IDENTIFICATION
C
      DIMENSION CVAL(3,1),STRS(1),INDS(1),ELEM(1)
      MASK = 77777B
      IF(NTEST.EQ.0) GO TO 1
      GO TO (115,2), NTEST
1  NCPI = 0
      ITS = 0
      NE1 = ELEM(1).AND.MASK
      NE2 = SHIFT(ELEM(1),45).AND.MASK

```

```

      NE2 = NE2 + NE1 - 1
      GO TO 100
2    NE1 = ELEM(1).AND.MASK
      NE2 = SHIFT(ELEM(1),45).AND.MASK
      NE2 = NE2 + NE1 - 1
      GO TO 101
100  NCP1 = NCP1 + 1
      IF(NCP1.GT.NOELM) GO TO 1000
      NINTR = CVAL(1,NCP1).AND.MASK
101  IF(NINTR.GT.NE2) GO TO 2000
      JUNK = NINTR - NE1 + 2
      NPNT = ELEM(JUNK).AND.MASK
      IF((SHIFT(ELEM(JUNK),6).AND.77B).NE.1) GO TO 8001
C
C      EXTRACT ROD STRESSES AND SET PROPERTIES
C
      JUNK = NINTR
      NPOD = INDS(JUNK).AND.27777B
      NPOD = NPOD+3*(LCS-1)
      NPRT = SHIFT(INDS(JUNK),46).AND.1777B
115  IF(NPRT.NE.NSBLK) GO TO 3000
      PCK = ELEM(NPNT+2)
      IF(STRS(NPOD).LT.0.) ELEM(NPNT+2)=ELEM(NPNT+3)=CVAL(2,NCP1)
      IF(STRS(NPOD).GT.0.) ELEM(NPNT+2)=ELEM(NPNT+3)=CVAL(3,NCP1)
      IF(PCK.NE.ELEM(NPNT+2)) ITS = 1
      GO TO 100
C
C      WE ARE COMPLETE
C
1000 NTEST = 3
      IF(ITS.EQ.0) NTEST = 4
      GO TO 9999
C
C      NEW ELEMENT MATRIX PARTITION IS NEEDED
C
2000 NTEST = 2
      GO TO 9999
C
C      NEW STRESS MATRIX PARTITION IS NEEDED
C
3000 NTEST = 1
      GO TO 9999
C
8001 PRINT 9001,NINTR
9001 FORMAT(//5X,* WARNING - INTERNAL ELEMENT*,I6,* IS NOT A ROD*/)
      GO TO 100
C
9999 CONTINUE
      RETURN
      END

```

GLOSSARY

The intent of this glossary is to provide a convenient reference for the common terms closely associated with the use of ATLAS.

Algorithm -- A set of rules or processes for the solution of a problem in a finite number of steps.

Alphanumeric Text -- A sequence of characters. An input data text string, which may not include # (a quotation mark when using a terminal), is initiated by the two characters ** and is terminated by the # character.

Alphanumeric Word -- A word with at least one nonnumeric character that can not be interpreted as a "decimal" or an "integer" number.

Analysis Reference Frame -- The GLOBAL or a local coordinate system used to measure the kinematics of a node. The input frame and the analysis frame associated with a node may be different.

ATlist -- This word denotes that one item, a list of data items, or a data generation list may be input for the corresponding input data. Any combination of the following ATlist options can be used when indicated within a data record:

$$\text{ATlist} = \left\{ \begin{array}{l} \text{item} \\ \text{list} \\ \text{a TO b <BY c>} \end{array} \right\}$$

Examples of ATlist are:

2 TO 10 BY 2	denotes	2,4,6,8,10
3,5,N2 TO N4,12	denotes	3,5,N2,N3,N4,12
A26,T1A05 TO T1A11 BY 2	denotes	A26,T1A05,T1A07,T1A09,T1A11

Boundary Condition (BC) Data -- Six degrees of freedom define the kinematic (response) of each node. The BC input data are used to identify one of the following activities for each nodal degree of freedom:

- a) FREE -- Unconstrained freedoms that are reduced from the gross (overall) stiffness equilibrium equations during the generation of reduced matrices.
- b) RETAIN -- Unconstrained freedoms that are retained in the reduced matrices. A freedom must be retained if
 1. It is to be used explicitly in defining the equations for a vibration analysis or a buckling analysis.
 2. It is a freedom of one substructure that interacts (joins structurally and kinematically) with a corresponding freedom in another substructure.
- c) SUPPORT -- Freedoms for which displacements are specified. A nonzero specified displacement is enforced or the freedom is rigidly fixed.

CATlist -- This word denotes that one execution parameter, a list of parameters, or a parameter generation list may be input. Any combination of the following CATlist options can be used when indicated within a parameter list for an ATLAS execution-control statement:

$$\text{CATlist} = \left\{ \begin{array}{l} a \\ (\text{list}) \\ a \text{ TO } b \text{ <BY } c \end{array} \right\}$$

Example uses of CATlist are:

```
EXECUTE STRESS (LC=(CRUISE,10,T5 TO T20,DIVE))
PRINT INPUT (INTERACT,SS=(2,5,4 TO 12 BY 2))
PRINT OUTPUT (REACTIONS,LC=(UP,D5 TO D40 BY 5))
```

Checkpoint -- A point during execution of a program at which processing is halted momentarily. Information associated with the job are saved for subsequent restart of the program execution.

Compile -- To produce a binary-coded (machine language) program from a program that is written in a source (symbolic) language.

Compiler -- A program that translates a programming language meaningful to a person into codes that are meaningful to a computer.

Control Card -- A directive to the computer operating system that describes requirements for a job and initializes execution of a program.

Control Command -- An ATLAS execution directive that is specified interactively to direct on-line execution of any ATLAS System Module and to control data management functions.

Control Deck -- See Execution Control Deck

Control Procedure -- A catalog (sequence) of ATLAS control statements that is used to perform a certain analytical function. Execution of a control procedure is initiated by a "PERFORM Procedure" control statement, where "Procedure" is the name assigned to the catalog. Standard ATLAS procedures are provided to perform standard types of analyses.

Control Program -- The ATLAS executive program which is created automatically by compiling the user-supplied ATLAS Execution Control Deck. The Control Program directs execution of ATLAS for solving structural analysis and design problems.

Control Statement -- An ATLAS execution directive that is defined in the user-oriented ATLAS language. Multiple control statements are used to create an ATLAS Control Deck; each statement initiates one or more execution steps in the problem solution.

Data Base -- A collection of interrelated physical data arranged in a unified manner that are serviced by a single data base manager.

Data Base Manager -- The hardware and software complex used to store and access data according to a set of rules for the definition, organization, protection, and efficiency of the data base.

Data Deck -- One or more data sets used to define a mathematical problem that is to be solved by ATLAS.

Data Item -- Either a number or an alphanumeric word (character string).

Data Management -- The organizing, cataloging, locating, storing, retrieving, and maintaining of data.

Data Record -- One or more input data items. A record commonly implies a data card, although it may require two or more cards. All data items in a data record can be input in a free-field format.

Data Set -- A block of input data that is associated with a particular engineering technology and is identified by a key-word. Examples are NODAL, BC, STIFFNESS, LOADS, MASS, and FLUTTER. A data set may include one or more input data subsets.

Data Subset -- A logical group of data within a data set. A data subset (and a data set) is composed of many data records.

Decimal Number -- A positive or negative number input either with a decimal point or by use of the FORTRAN E-Format.

Design CYCLE -- Each automated design cycle (iteration) includes the following:

- a) Generation of the structural stiffness matrices
- b) Solution for nodal displacements and element stresses
- c) Calculation of margins of safety based on the specified design criteria
- d) Updating (resizing) the element properties

Diagnostic -- A message that results from the detection of an error or malfunction in the computer hardware or software during execution of a job.

Direct Access -- Synonymous with random access.

Dump -- To copy the contents of all or part of the computer core memory onto an external storage medium.

Element Reference Frame -- A coordinate system associated with a finite element. Each structural finite element and some of the mass finite elements have their own element reference frames. Each frame is a right-handed, rectangular system that is oriented according to the sequence of nodes used to define the element. Input section properties, calculated stresses, and optionally, the distributed element loads, are based on the orientation of these reference frames.

ERROR -- Whenever a computer operating system error occurs during execution of ATLAS, or when a fatal inconsistency is detected either in the data or in the user-selected execution logic, an ERROR message is printed. In this case, only the execution directives included in an "ERROR PROCEDURE" within the Control Program are processed prior to terminating the job.

Execution Control Deck -- A sequence of user-specified ATLAS execution-control statements that identifies which ATLAS functions are to be used in solving a problem.

Execution Parameter -- A user-selected option associated with execution of an ATLAS program module. A parameter can identify a single value, a list of values enclosed by parentheses, or generation of parameter values using the "a TO b <BY c>" option. Example parameters are:

```
SET=2,LC=(10,12,40),STRESSES  
STIF=KRED,STURM,SUBSETS=N2 TO N6
```

Executive -- That software which manages other software to perform data management or numerical computations.

Extract -- To choose from a set of items all those that meet some criteria.

Filename -- The name of an ATLAS System data file. All input data are stored in the file DATARNF, whereas output data from a processor are stored in its own file. Examples are STIFRNF for the STIFFNESS Processor, DESIRNF for DESIGN, LOADRNF for LOADS, and STREARNF for STRESS.

FORTRAN -- FORMula TRANslating system. A programming language used to write programs in a form that resembles algebra.

FREE -- See Boundary Condition (BC) Data

GLOBAL Reference Frame -- The common coordinate system that is used for referencing input data and output data associated with a structural and/or a mass model. This coordinate system is a right-handed, Cartesian triad denoted by X-Y-Z.

Hard Copy -- A printed copy of machine output in a visually readable form (e.g., listings, printed reports, plots displayed on paper).

Inactive Freedom -- A nodal degree of freedom that has zero stiffness and is automatically ignored by ATLAS during the solution process.

Input Reference Frame -- The GLOBAL or a local coordinate system used to define nodal coordinates.

Integer Number -- A positive or negative number without a decimal point.

Integrated Software System -- A network of program modules for performing single-disciplined or multidisciplined tasks with a common data base manager, and with standard user interfaces for execution control and data base interrogation.

Interact Tree -- A diagram defining how substructures are interacted (merged) to form the total structural model for substructured analyses (see Substructure) .

Interfaced Programs -- Independently-developed standalone programs that are interfaced by data converters.

Interpreter -- A program that translates and executes user-oriented language statements at run time.

Job -- A specified group of tasks prescribed as a unit of work for a computer. A job usually includes a program, instructions to the operating system, and the necessary data.

Job Control Deck -- A sequence of job control cards required for executing a job deck on a computer.

Job Priority -- A class assigned to a job that determines the priority to be used by the computer operator in scheduling and allocating resources for execution of the job.

Key-Word -- A data item or an execution parameter that must be input as shown or at least the underlined portion thereof.

Library Routine -- A routine that is kept permanently available and which is designed to accomplish some commonly used mathematical function.

Listing -- A printout of computer codes and data.

Load -- To enter data from external storage into memory storage or working registers of a computer.

Local Reference Frame -- A right-handed, orthogonal coordinate system that may be rectangular, cylindrical or spherical. All local frames are defined relative to the GLOBAL reference frame as part of the NODAL data set. Local frames can be used for the following purposes:

- a) Define node locations

- b) Define nodal load directions
- c) Define nodal boundary conditions and support displacements
- d) Define geometry components of the structural and/or mass model
- e) Measure the nodal displacement components resulting from static-equivalent loads or the mode shape components associated with structural buckling or vibration
- f) Measure the input and output concentrated-mass moments of inertia

Mass Distribution Condition -- A user-selected combination of any of the following types of mass-related model components for calculating reduced mass matrices associated with the retained nodal freedoms or for calculating panel-weight matrices for a SET:

- a) Structural mass defined by the stiffness elements
- b) Non-structural mass defined by the mass finite elements
- c) Fuel distributions
- d) Payload distributions
- e) Concentrated masses

Material Codes -- Three types of linearly elastic materials can be used in defining the structural finite element model.

- a) Standard -- Commonly-used, isotropic materials that can be accessed directly. Their reference codes are M1 through M10.
- b) Special -- Three-dimensional user-defined materials that can be isotropic or orthotropic. Their reference codes are M51 through M99.
- c) Composite -- Two-dimensional user-defined orthotropic materials for analyses using the composite finite elements. Their reference codes are C1 through C31.

The data corresponding to each material code include its density and the following information, generally for a number of temperature levels.

- a) Young's modulus
- b) Poisson's ratio
- c) Shear modulus
- d) Thermal strain
- e) Ultimate and yield stress allowables.

Mathematical Model -- The system of mathematical equations that characterize the problem defined by the model-definition data deck. Solutions of the equations are performed by ATLAS.

Merge -- To combine items from two or more similarly ordered sets into one set that is arranged in the same order.

Mid-Surface Nodes -- Nodes with four coordinates used primarily for defining the ATLAS SPAR, COVER, and CCOVER finite elements.

MODE1 Data Input -- The end of a data record is identified either by the / symbol or by the \$ symbol. Information following the / symbol are treated as data deck comments, whereas information following the \$ symbol are assumed to be data items for the next data record.

MODE2 Data Input -- The same as MODE1 Data Input except the right-hand boundary of a card also acts as an end of data record, and a plus sign (+) must be used as the last character on a card when the record continues onto the next card.

Module -- A computer program that performs a well-defined engineering, mathematical, or data management task.

Node -- A point in space used to idealize the geometry of a real structure. Each node is identified by an integer number. Structural characteristics are idealized by finite elements that are interconnected at the Structural nodes. A node used to orient and/or position a finite element relative to Structural nodes is called an Auxiliary node. A node used to define mass finite elements and concentrated masses is called a Mass node.

Off-Line -- Pertaining to the operation of peripheral equipment that is not under the control of the computer.

On-Line -- Pertaining to the operation of peripheral equipment that is under the control of the computer.

Operating System -- An integrated collection of routines that provides a logical extension of the computing hardware. It controls the execution of computer programs and may provide input/output control, accounting, compilation, storage assignment, etc.

Output -- Information transferred from the computer to a peripheral device.

Overlay Program -- A program that is comprised of modules (subprograms) with segments that use the same storage locations at different times during execution. When one routine is no longer needed in storage, another routine replaces all or part of it.

Pivot Decay Number -- The number of significant digits of accuracy lost during the solution of the stiffness equilibrium equations by the CHOLESKY processor. Generally, large decay numbers indicate an ill-conditioned mathematical model which should not be accepted for further analysis. This criterion is used to identify local mechanisms or regions of weak stiffnesses inherent to the structural model.

Postprocessor -- A program that extracts, formats, and generates print/plot displays of data (input and results), or a program that stores (saves) data to be used at a later time. Each ATLAS postprocessor is identified by a key-word. Examples are NODAL, FREEBODY, GRAPHICS, STRESS, and FLUTTER.

Preprocessor -- A program that reads, decodes, generates, interrogates, and formats data for another program. Each ATLAS preprocessor is identified by a key-word. Examples are NODAL, STIFFNESS, LOADS, and MASS.

Problem-Oriented Language --A language designed for convenient definition of a technical engineering problem for solution by a computer program.

Procedure -- A block (sequence) of statements that defines the course of action to be taken for the solution of a problem.

Processor -- A program that performs technical analytical computations. Each ATLAS processor is identified by a key-word. Examples are STIFFNESS, DESIGN, LOADS, STRESS, MASS, VIBRATION, and FLUTTER.

Random Access -- Pertaining to a storage device in which the accessibility of data is effectively independent of the location of data.

Reference Frame -- See Local Reference Frame

RETAIN -- See Boundary Condition (BC) Data

Savefile -- The name of an ATLAS System data file that is used for checkpoint/restart of job execution. The ATLAS savefiles are identified by the following key-words: SAVESSF, SAVESS1, SAVESS2, SAVESS3, and SAVESS4.

SET -- A structural and/or mass model that is identified in the input data and the output data by an integer number in the range 1 through 36. A SET, as defined by the model-definition data, is limited to the following maximum quantities.

- a) 4095 nodes
- b) 32 767 structural finite elements
- c) 32 767 mass finite elements
- d) 10 boundary condition STAGES
- e) No explicit limit on the number of loadcases
- f) 100 mass distribution conditions

SNARK -- A high-level, matrix-oriented language that is used by the ATLAS System and is available for creating an ATLAS Control Program to perform the following:

- a) Algebraic matrix operations

- b) Input and output of data matrices on either random or sequential files
- c) Manage storage of data in the computer core memory
- d) Manipulate data in the ATLAS data base

Detailed descriptions are presented in the SNARK User's Manual ATLAS documentation.

Software -- A set of computer programs, procedures, and associated documentation concerned with the operation of a data processing system.

STAGE -- A group of boundary condition (BC) constraints used in solving the structural equations of equilibrium. A maximum of 10 STAGES can be defined for each SET.

Statement -- An expression or instruction that is written in a programming language.

Subsets -- User-selected nodes, elements, and data components that are to be extracted and manipulated by ATLAS. The types of subsets that can be defined by a SUBSET DEFINITION data set are:

- a) NODAL -- Selected nodes. Coordinates, node loads, displacements, reactions, and lumped masses can be extracted for node subsets. Ordered node subsets are used to establish a particular sequence of nodes.
- b) STIFFNESS -- Selected stiffness finite elements. Element-definition data, stresses, and distributed loads, for example, can be extracted for element subsets.
- c) MASS -- Selected mass finite elements.
- d) LABELS -- Selected data component labels. The input and analysis data values corresponding to the selected labels are extracted for subsequent graphical or print display.

Substructure -- A part of the total structural model that is idealized separately and is identified as a SET. Multiple substructures are interacted (joined together) to form the total structural model. The following maximum quantities apply to substructured stress and dynamic analyses.

- a) 999 unique substructures
- b) 28 substructures interacting to form a next higher-level substructure
- c) No explicit limit on the number of interact levels

SUPPORT -- See Boundary Condition (BC) Data

SUPSTAGE -- A superposition loadcase that is defined as a linear combination of other loadcases for superimposing displacements and stresses.

Terminal -- A peripheral device through which data can either enter or leave the computer.

Timesharing -- The allocation of computer time and computer services to multiple users so that a number of programs are processed simultaneously.

User -- Anyone who requires the services of a computing system.

User Matrix -- An ATLAS data matrix whose name can be specified by the user during problem solving for greater versatility of data management.

User Oriented Language -- A programming language designed for convenient expression of steps to be used in the solution of a wide class of problems.

WARNING -- Whenever ATLAS detects an ambiguity in the data that can be resolved without user interaction, a WARNING message is printed and processing of the job continues without interruption.

REFERENCES

- 5-1. Clough, R. W., and C. A. Felippa, "A Refined Quadrilateral Element for Analysis of Plate Bending," 2nd Conference on Matrix Methods in Structural Mechanics, AFFDL-TR-68-150, 1968.
- 5-2. Garvey, S. J., "The Quadrilateral Shear Panel," Aircraft Engineering, 1951.
- 5-3. Guyan, R. J., "Reduction of Stiffness and Mass Matrices," J. Am. Inst. Aeronautics and Astronautics, 1965.
- 5-4. Archer, J. S., "Consistent Mass Matrix for Distributed Mass Systems," J. Structural Division, Proceedings ASCE, 1963.
- 5-5. Hink, G. R., et al., "A Method for Predicting the Stability Characteristics of an Elastic Airplane," Vol. III--FLEXSTAB Program Manual, NASA CR-114714, Ames Research Center, Oct. 1974.
- 5-6. Theodorsen, T., "General Theory of Aerodynamic Instability and the Mechanism of Flutter," NACA TR-496, 1934.
- 5-7. Richmond, L. D., "A Rational Method of Obtaining Three-Dimensional Unsteady Aerodynamic Derivatives of Intersecting Airfoils in Subsonic Flow," D6-7401, Boeing Commercial Airplane Company, April 1962.
- 5-8. Theodorsen, T., and I. E. Garrick, "Nonstationary Flow about a Wing-Aileron-Tab Combination Including Aerodynamic Balance," NACA TR-736, 1942.
- 5-9. Giesing, J. P., Kalman, T. P., and W. P. Rodden, "Subsonic Unsteady Aerodynamics for General Configurations," Part I, Vols. I and II, AFFDL-TR-71-5, Feb. 1971.
- 5-10. Ii, J. M., and D. F. Tankersley, "Unsteady Aerodynamic Interaction for Advanced Aircraft Configurations in Subsonic Flow (Improved Doublet-Lattice TEV 155)," D6-41258, Boeing Commercial Airplane Company, Dec. 1973.

- 5-11. Ii, J. M., Borland, C. J., and J. R. Hogley, "Prediction of Unsteady Aerodynamic Loadings of Non-Planar Wings and Wing-Tail Configurations in Supersonic Flow," Part I--Theoretical Development, Program Usage and Application, AFFDL-TR-71-108, Wright-Patterson Air Force Base, Ohio, March 1972.
- 5-12. Rowe, W. S., Winther, R. A., and M. C. Redman, "Prediction of Unsteady Aerodynamic Loadings Caused by Trailing Edge Control Surface Motions in Subsonic Compressible Flow," NASA CR-2003, June 1972.
- 5-13. Desmarais, R. N., and R. M. Bennett, "An Automated Procedure for Computing Flutter Eigenvalues," AIAA Paper No. 73-393, presented at the AIAA/ASME/SAE 14th Structures, Structural Dynamics and Materials Conference, Williamsburg, Virginia, March 20-22, 1973.
- 5-14. Hill, R., "The Mathematical Theory of Plasticity," Oxford University Press, 1960.
- 5-15. Adelman, H. W., et al., "An Improved Method for Optimum Design of Mechanically and Thermally Loaded Structures," NASA TN D-7965, August, 1975.
- 5-16. Zoutendijk, G. G., "Methods of Feasible Directions," Elsevier, New York, 1960.
- 9-1. Hink, G. R., et al., "A Method for Predicting the Stability Characteristics of an Elastic Airplane," Vol. III--FLEXSTAB Program Manual, NASA CR-114714, Ames Research Center, Oct. 1974.
- 9-2. Craidon, C. B., "Description of a Digital Computer Program for Airplane Configuration Plots," NASA TM X-2074, Sept. 1970.
- 9-3. McCormick, C. W., "The NASTRAN User's Manual," NASA SP-220(01), June, 1972.

1. Report No NASA CR-159041		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle ATLAS--AN INTEGRATED STRUCTURAL ANALYSIS AND DESIGN SYSTEM, "ATLAS User's Guide"				5. Report Date July 1979	
				6. Performing Organization Code	
7. Author(s) Dr. Rodney L. Dreisbach, Editor				8. Performing Organization Report No. D6-25400-0001	
9. Performing Organization Name and Address Boeing Commercial Airplane Company (BCAC) P. O. Box 3707 Seattle, Washington 98124				10. Work Unit No.	
				11. Contract or Grant No. NAS1-12911	
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration (NASA) Washington, D.C. 20546				13. Type of Report and Period Covered Contractor Report	
				14. Sponsoring Agency Code	
15. Supplementary Notes Program Managers: Dr. Gary L. Giles (NASA Langley Research Center); Dr. Ralph E. Miller, Jr. and Mr. Kenneth H. Dickenson (BCAC). This document is one volume of the following series of documents describing the ATLAS version 4.0 system: CR-159041--ATLAS User's Guide; CR-159042--System Design Document; CR-159043--User's Manual (Input and Execution Data); CR-159044--Random Access File Catalog; CR-159045--System Demonstration Problems; CR-159046--Design Module Theory.					
16. Abstract This document describes some of the many analytical capabilities provided by the ATLAS integrated structural analysis and design system. The example data and fundamental technical considerations presented in this guide are intended to help orient the engineer in using ATLAS, and to direct him in using the other ATLAS System documentation. ATLAS is operational on the Control Data Corporation (CDC) 6600/CYBER computers in the batch, on-line, and interactive modes. It is a modular system of computer codes with common executive and data-base management components. The system provides an extensive set of general-purpose technical programs with analytical capabilities including stiffness, stress, loads, mass, substructuring, strength design, unsteady aerodynamics, vibration, and flutter analyses. The sequence and mode of execution of selected program modules are controlled by a common user-oriented language. Execution of selected modules with external interfaced programs is supported by the ATLAS data-base and ATLAS data manager. Interactive user interfaces are provided for performing execution-control, data-file editing, and graphical-display of selected data. Model-definition input data are written in a problem-oriented language. Input-data generation options and input-data checks provided by the preprocessors minimize the amount of data and flowtime for problem definition/verification. Post-processors allow selected data to be extracted, manipulated, and displayed via on-line or off-line prints and plots for monitoring and verifying problem solutions.					
17. Key Words (Suggested by Author(s)) Integrated system, structural analysis, automated design, stiffness, stress, loads, weights, dynamics, aerodynamics, flutter, aero-elasticity, finite elements, interactive graphics, preprocessing, postprocessing, computer-aided design			18. Distribution Statement Unclassified - Unlimited		
19. Security Classif. (of this report) Unclassified		20. Security Classif. (of this page) Unclassified		22. Price*	